Analysis of the Thermal Indoor Climate with Computational Fluid Dynamics for Buildings in Sub-arctic Regions

Daniel Risberg
Energy Engineering
DOCTORAL THESIS

Analysis of the Thermal Indoor Climate with Computational Fluid Dynamics for Buildings in Sub-arctic Regions

Daniel Risberg
Energy Engineering
Division of Energy Science
Department of Engineering Science
Luleå University of Technology
SE-97187 Luleå Sweden.
Abstract

This thesis aims to increase the knowledge of simulation of thermal indoor climate for nearly zero energy buildings in a sub-arctic climate. Air heating systems in cold climate generate temperature gradients, which negatively affects the thermal indoor climate. Standard multi-zone modeling has problems with predicting these gradients.

In this work, Computerized Fluid Dynamics (CFD) simulations are used to model the temperature gradients. The consequences of reducing the cell sizes for the simulation volume are estimated and case studies of different building and heating systems are presented. The CFD method is validated for a traditional underfloor heating system and also for an air heating system.

Furthermore, the effects of snow on heat losses for common building foundations are investigated, and snow is shown to be an important boundary for CFD simulations of a building. The snow and ground freezing are shown to reduce the annual heat losses between 7-10%.

The CFD method is shown to be a suitable method for predicting thermal indoor climate. The method can determine the temperature variations inside a building, for different rooms, floors and heating systems. The CFD method is most appropriate for local distributed systems. For traditional hydronic systems the method may be overambitious, since a good indoor climate is usually achieved anyway.

Heat transfer coefficients are inaccurate when calculated using standard wall functions used in many turbulence models (like the k-ε model) for surfaces with a high heat transfer rate and natural convection. Automatic wall functions have shown better accuracy for this type of problem, but they require more cells. In order to still use the k-ε model, a user defined wall function is investigated. This method gave good results and a significant reduction in the number of necessary cells in the simulation volume. The validation of the indoor climate shows that the wall boundary conditions are important for predicting the indoor temperature variations for steady state simulations.

New buildings have a higher thermal inertia, which affects the heat losses. It is important to include this effect in the boundary condition calculations for a CFD model.

The CFD simulations show that air heating and local distributed heating systems have difficulties in fulfilling a good thermal comfort inside all rooms. This is especially true for rooms with exhaust air and closed doors and multi-story buildings. Results from a CFD simulation can be used to improve the thermal comfort in these cases.
Acknowledgements

The author gratefully acknowledges the Swedish Governmental Agency for Innovation Systems (VINNOVA) within the Attract project and the INTERREG Nord program for the founding of my work.

The authors would also like to acknowledge NCC Construction Company and Abelco for their cooperation. I also want to show my appreciation to house manufactures in Norrbotten and other companies and partners involved in the different projects.

I would thank everyone that in one way or another has helped, inspired and motivated me during this work. Especially my principal supervisor Lars Westerlund, my co-supervisor Erik Elfgren and the co-authors Mikael Risberg, Mattias Vesterlund and Jan Dahl.

Furthermore, thanks to my colleges at Division of energy engineering at Luleå University of Technology for creating a good work environment and friendly atmosphere. Extra thanks to Elizabeth Wetterlund, Andrea Toffolo and Petter Lundkvist for valuable discussions and feedback.

Finally, I want to give a lot of thanks to my girlfriend Lina and other members in my family for their support and always showing an interest of my work.
Appended papers


II. Risberg, D. Risberg, M. Westerlund, L. CFD modelling of radiators in buildings with user-defined wall functions, Applied thermal engineering 2016. Vol. 64, s. 266-273


IV. Risberg, D. Risberg, M. Westerlund, L. Investigation of thermal indoor climate for a passive house in a sub-Arctic region using computational fluid dynamics. Accepted in Indoor and built environment.


Papers not included in the thesis

Author contribution

Paper I and II
D. Risberg planned and performed the simulations, calculations and wrote the paper with support by the co-authors.

Paper III
D. Risberg planned and performed the simulations. The paper was jointly written by D. Risberg and M. Vesterlund, with assistance from L. Westerlund and J. Dahl.

Paper IV
D. Risberg planned and performed the simulations, calculations and measurements with support by the co-authors. All authors jointly analyzed the results and summarized the conclusions in the article. All authors read, modified, commented and approved the manuscript.

Paper V
D. Risberg planned and performed the simulations and calculations with support by the co-authors. D. Risberg wrote the paper with assistance of L. Westerlund and M. Risberg.
# Table of Contents

1 Introduction...................................................................................................................... 1
   1.1 Research aim and objectives ....................................................................................... 2
   1.2 Overview of appended papers .................................................................................... 2
2 Literature overview........................................................................................................ 5
   2.1 Building thermal system ............................................................................................ 5
      2.1.1 Building elements and materials ....................................................................... 6
      2.1.2 Heat transfer processes ....................................................................................... 8
      2.1.3 Outdoor conditions ............................................................................................. 9
      2.1.4 Internal heat ...................................................................................................... 9
   2.2 Energy usages in buildings ......................................................................................... 11
      2.2.1 Operational energy ........................................................................................... 11
      2.2.2 Embodied energy ............................................................................................. 12
      2.2.3 Low energy buildings in Sweden ..................................................................... 12
   2.3 Thermal comfort ........................................................................................................ 15
   2.4 Simulation methods ................................................................................................... 18
      2.4.1 Multi-zone Method ........................................................................................... 18
      2.4.2 Computational Fluid Dynamics ......................................................................... 19
      2.4.3 Zonal Method .................................................................................................. 19
      2.4.4 Coupled multi-zone and CFD simulations ....................................................... 20
   2.5 CFD in buildings ..................................................................................................... 20
      2.5.1 Low energy buildings and Passive house studies ............................................. 22
   2.6 Summary of literature and motivation ..................................................................... 23
3 Studied systems.................................................................................................................. 25
   3.1 Studied Buildings ..................................................................................................... 25
   3.2 Foundations ............................................................................................................. 28
4 Validation procedure....................................................................................................... 31
   4.1 Validation of Building 1 .......................................................................................... 31
   4.2 Validation of Building 3 .......................................................................................... 32
1 Introduction

The building stock accounts for 41% of the total energy use in the European Union [1]. This number is in line with the Swedish usage of around 40% (143TWh, 2015) for buildings and service. Households and facilities account for around 90% of energy use in this sector [2]. The European Union’s goal is to reduce the greenhouse gas emissions by 40% to 2030 compared to 1990’s level [3]. The energy supply to buildings in Sweden has a low amount of fossil fuel and a high amount of electricity and district heating. Although the extent of greenhouse gases from heating of buildings is low, a reduction in energy demand in the sector will release electricity to other sectors with a high amount of fossil fuel, for example the transport sector.

The goal on the national level in Sweden is to reduce the energy demand in buildings by 50% until 2050 compared to 1995’s level [4]. In order to accomplish that goal an implantation of near zero energy houses by 2020 will be added. Many constructors want to build even better than the requirements; several new buildings are adapted to different low energy concepts. Classifications like the passive house concept and Sweden’s Green Building Council classification for environmental buildings are increasing. The buildings are more insulated and have a higher investment cost compared to standard buildings; the decreased demand of supplied heat entails investigation of cheaper heating systems like air distributed heat and local heat sources as an alternative to the traditional heating systems.

The different classifications also have a higher demand on the thermal indoor climate, which induces increased simulations of the thermal comfort in buildings. Through simulations new ideas and technologies can be investigated before introduction in reality. Common simulation methods like the Multi Zone approach [5] used in simulation software like IDA ICE [6], Energy PLUS [7] and TrnSys [8], have problems with predicting airflow and heat flow between different rooms, due to the single node approach and well mixed air assumption. These software packages are not optimal for thermal indoor climate simulations.

Computational fluid dynamics (CFD) is a more sophisticated tool for this type of simulations where all external parameters for describing the thermal climate are included. The main problem for this field of application is long simulation times, since the computational volume usually is large. Wall functions for coarse grids have problems with predicting the heat transfer from walls because of the treatment of the natural convective fluid flow. The equations solved by CFD for the fluid flow also requires short time steps compared to the
Multi Zone simulations. Steady state simulation is less time demanding and more reasonable to use for the CFD simulations. Due to the steady state approach, boundary data for heat losses are an important input parameter for the CFD simulation.

The Swedish town of Kiruna is located in the northern part of the country and an urban transformation is to take place due to an extension of the mine. Due to the lack of classified houses in this region, the northernmost passive house is located in Umeå and a lot of new production of houses makes this place a good location for case studies. Kiruna is located in sub-arctic climate above the Arctic Circle with an average annual outdoor temperature of -1.5°C and temperatures below -20°C is very common. Heating systems that are able to provide a good thermal comfort in this region should also be suitable for other locations with warmer climate in the country.

1.1 Research aim and objectives

The overall aim of the thesis is to improve the knowledge about simulation of thermal indoor climate for buildings in cold climate and its application for computer based simulations. The work is done in order to simplify the use of CFD as a tool in order to model the thermal climate, especially for air heating systems. The models must be simplified in order to reduce the simulation time.

The specific objectives of this thesis are as follows.

- Identify important parameters and simplifications for CFD studies of thermal climate.
- Develop a CFD model that can describe the indoor climate in low energy buildings with reasonable simulation time.
- Investigate the thermal indoor climate for a whole building in cold climate with different heating systems using CFD.
- Evaluate the indoor climate simulation for an air heating system.
- Investigate whether common simulation methods for heat losses should be modified in order to take into account the effects of snow cover.

Table 1 summarizes these objectives and how they are related to the papers in the thesis.

1.2 Overview of appended papers

The work in this thesis considers simulations of the indoor climate and energy subsystems for building applications related to cold climate. Boundary conditions and simulation setup was investigated in order to improve the model performance. Different buildings and heating systems were studied in order to describe the thermal indoor climate more reliably and with less simulation time.
A CFD model of a two-room setup was investigated in Paper I. The influence of grid size with grid convergence index and the most common RANS turbulence models was studied. Vertical temperature gradients during annual mean outdoor temperature and a winter case were compared. The model was scaled up to a complete building and validated for an underfloor heating system.

From the results in Paper I, the \( k-\varepsilon \) model with scalable wall treatment was chosen; this turbulence model exhibits problems with treating natural convective heat transfer, while other turbulence models have shown to be too computationally demanding. Paper II studies how to deal with natural convective heat transfer for a radiator in order to simplify the simulations, thus reducing the numbers of cells and the simulation time. By adding user-defined wall functions, the number of cells can be reduced considerably compared to automatic wall treatment. The user-defined wall function proposed can also be used with a correction factor for different radiator types without the need to resolve the radiator surface in detail.

To explore the viability of the approach in Paper I, the thermal indoor climate in a building was studied considering three different heating systems: (1) an underfloor heating system, (2) air heating through the ventilation system and also (3) an air/air heat pump installation for the whole one story building in Paper III. The indoor climate was evaluated according to recommendations to achieve a tolerable indoor climate, as required by the Swedish authorities.

Heating a building through the ventilation system was simulated and validated in detail in Paper IV. This was done in order to investigate if local distribution of heat can be simulated accurately with CFD for a two-story building. The model was validated with temperature and velocity measurements. The paper also evaluates the thermal comfort (according to ISO 13370 standard) in possibly the northernmost passive house in Sweden.

The results in Paper IV have shown that the correct determination of the heat losses is important for indoor climate simulations. Classical building simulations (using the ISO 13370 standard) do not take into account the improved insulation from snow. In Paper V the influence of snow and ground freezing for common foundation constructions was investigated. The work was done in order to study the heat flux from different building foundation types during the winter with a 3D numerical model.
**Table 1. Objectives in the appended papers.**

<table>
<thead>
<tr>
<th>Appended papers</th>
<th>Objectives</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Paper I</strong> CFD-simulation of indoor climate in low energy buildings, Computational setup</td>
<td>Identify important parameters and simplifications for CFD studies of thermal climate.</td>
</tr>
<tr>
<td><strong>Paper II</strong> CFD modeling of radiators in buildings with user-defined wall functions</td>
<td>Develop a CFD model that can describe the thermal indoor climate in low energy buildings with reasonable simulation time.</td>
</tr>
<tr>
<td><strong>Paper III</strong> CFD simulation and evaluation of different heating systems installed in a low energy building located in sub-arctic climate</td>
<td>Investigate indoor climate for a whole building with different heating systems using CFD.</td>
</tr>
<tr>
<td><strong>Paper IV</strong> Investigation of thermal indoor climate for a passive house in a sub-Arctic region using CFD</td>
<td>Evaluate the indoor climate simulation for an air heating system.</td>
</tr>
<tr>
<td><strong>Paper V</strong> The impact of snow and soil freezing for commonly used foundation types in sub-arctic climate.</td>
<td>Investigate whether common foundation types should be modified in order to take into account the effects of snow cover.</td>
</tr>
</tbody>
</table>
2 Literature overview

This section covers the buildings overall thermal systems, simulation methods for the thermal indoor climate and energy simulations of buildings. This section also covers some earlier studies concerning CFD used for studying the indoor climate and motivations for this thesis.

2.1 Building thermal system

Indoor climate is described by four climate factors; thermal climate, indoor air quality, sound and light. The indoor air quality is the concentration of polluting gases and particles in the air [9], mainly effected by the polluting sources, outdoor conditions and the ventilation performance. Thermal climate is described as the heat balance of the human body. It is affected by all heat transfer mechanisms that appear against the human skin and the heat generated inside the body, in order to have a stable body core temperature around 37°C [10]. Fanger [11] has developed a model from an experiment describing the statistical number of dissatisfied people, where most of the heat transfer mechanisms that appear are considered. Parameters that affect the thermal comfort are the metabolic rate, external work, clothing insulation, air temperature, radiation temperature, air velocity and relative humidity [12].

The last four parameters are influenced by the envelope surfaces and heating system of the building. A simplified illustration is presented in Figure 1. A good thermal comfort competes with annual energy usage, heating cost, carbon dioxide emissions and construction/installation costs.

![Figure 1. Schematic view of the building's energy balance.](image)
The heat balance of a building is divided into four major parts: building elements and materials, heat transfer processes, internal heat and external conditions. In Figure 2 an overview of these parts and different subparts are presented.

Figure 2. Overview of parts included in the building heat balance.

2.1.1 Building elements and materials

The heat transfer through walls and ceiling results from building elements causing a high amount of heat losses for most houses due to their relative large area. These elements are thereby also the most insulated parts of a building. Their direct contact with the outdoor conditions makes a one dimensional (1D) approach applicable for building simulation of these building elements [13]. The use of overall heat transfer coefficients according to the ISO 6946 Standard [14] is applicable for heat demand calculations for walls and ceiling constructions. The most commonly used building energy simulation software (BES) uses a finite difference method and can be used to predict the transmission losses including thermal inertia of the building elements. The method has been validated according to the ISO 13791 standard [[15]-[16]].

The largest heat flux of the building is through the windows, and due to the high amount of window area in newly built houses it is often the building element with the highest heat losses. Major improvements have been made during the last decades in order to reduce the heat losses. Window glazing has been improved by coatings that reduce the emissivity, which can today be reduced to 0.013[17]. A triple glazing window with low emissivity
glasses and low solar heat gain coefficient can reach an overall heat transfer coefficient (U) value around 0.5 W/m² K for the glass part and a quadruple glazing window a U value of around 0.3 W/m² K [18]. In total including also the window frame this increases the U value of the window. Commercial triple glazing windows have a U-value of around 0.77 W/m² K [19] and quadruple glazing windows have a U-value of around 0.60 W/m² K [20]. Due to the high cost of low e glazing (low emissivity glazing) the state of the art material like aerogel and vacuum can replace some of the low e glasses [18].

Foundation heat losses are complicated to predict since many parameters affect, for example the high amount of thermal inertia in the soil [21]. In cold climate with temperatures below zero and snow these parameters also affect the thermal performance. The standard calculation method [22–24] can only handle fix geometry and the outdoor temperature requires sinusoidal time-varying outdoor temperature [25]. This makes numerical methods more suitable for foundation heat loss calculations. The common BES like IDA ICE use the method described in ISO 13370 standard. It is a 1D calculation method [22] while the problem is multidimensional [26]. The finite element method (FEM) is the most applicable method for this kind of problem. Janssen et al. studied the influence of the soil moisture [27] and the influence of soil freezing [28] using FEM. The snow was shown to have a large impact on soil coupled heat transfer [29].

Thermal bridges are a local area of the building where the heat resistance is lower than for the surrounding material. The thermal bridges are a consequence of the 1D approach with overall heat transfer coefficients in building physics that in reality is a 2D or 3D heat transfer problem [13]. The thermal bridges increase the heat losses through the building envelope surfaces. The thermal bridges are not specifically a building element but mainly appear in the joint between building elements, but also from frame structures and built-in constructions or installations that penetrate insulated elements. The thermal bridges can be decided with different methods, where the ISO 14683 standard overestimated the error with up to 50% and calculation formulas have errors of +/−20% [30]. By static numerical models Larbi [31] has shown that the errors become below 10%.

Building material is mainly specified by mechanical properties. It is often stated in terms of mechanical properties or thermal insulation capabilities. For heat transfer and thermal indoor climate performance the insulation capabilities have the highest interest. Command insulation materials are well studied and specified in the ISO standards [14,32,33]. Mohammad and Al-Homoud have summarized the thermal performance of the most common building insulation materials [34]. A lot of research studies with state of the art material have been made, like phase change materials [35], Vacuum insulation panels (VIP) [36], Gas-filled panels [37] and aerogels [38]. Also improvement of commonly used building materials (for example. Mineral wool, expanded polystyrene (EPS) Extruded polystyrene (XPS), Cellulose and Polyurethane (PUR)) or new material like nano insulation materials
and dynamic insulation materials [39] have been studied. Overall all new building materials are far more expensive than the standard building materials for the same level of thermal performance, Jelle show the breakeven for VIP where land cost for the increased living area is included [39].

2.1.2 Heat transfer processes

The heat transfer processes that appear for the building elements are the classical heat transfer mechanisms: conduction, convection and radiation. A building also has ventilation that is a controlled flow of fresh outdoor air to the indoor environment and infiltration that is an uncontrolled ventilation flow. For these parameters many studies have been conducted in order to predict the building heat losses in the best way. The heat conduction and radiation are treated by the classical heat transfer analogy. The solar radiation is often calculated from a weather database included in the software for building simulations.

The convective heat transfer sensitivity to different flows and buoyancy makes it the weakest link in the overall heat transfer approach [40]. Convection coefficients for indoor surfaces have been performed in several studies summarized by Peeters et al. [40]. Many experimental studies of external convective heat transfer coefficients are also performed; some of them are summarized by Palyvos et al. [41].

The McAdams formula for external heat transfer coefficients is the most commonly used in BES software packages [42]. Mirsadeghi et al. [42] also showed that the result could differ ±30% for cooling demand and ±6% for the heating demand for a cubic building with a U-value of 0.4 W/m² K between different formulas. For more insulated buildings the effect of convective heat transfer coefficients both for the internal and external surfaces is reduced, since the conduction resistance through the building surfaces increases.

The infiltration driving force is the pressure difference between the outdoor and indoor air. The pressure difference is influenced mainly by the wind pressure and stack effect, where the stack pressure is a function of the ambient air temperatures (outdoor and indoor) and height of the building [43]. The infiltration rate is also highly dependent on the planning and construction of the building. It is hard to predict the infiltration rate and it is often measured by a pressure test with 50 Pa under pressure inside the building. The simplest way to threat infiltration is established by Liddament [44]. The method takes the measured airflow at 50 Pa and divides it by 20 to calculate the average infiltration rate. This method does not take into account the location of the building, wind conditions and temperatures. Younes et al. [43] describe some more advanced empirical techniques and theoretical methods for treating infiltration in buildings where CFD is one of the most advanced techniques.

Ventilation affects the heat loss and energy use of the building. It is also important from an indoor climate perspective, mainly for indoor air quality but also for the thermal comfort. Ventilation, i.e. natural ventilation where outdoor wind conditions and buoyancy forces...
inside the building achieve a ventilation flow due to the pressure difference (based on the same driving forces as infiltration), or mechanical ventilation where fans make a pressure difference. The mechanical ventilation can be either a supply/exhaust system or only an exhaust system. For mechanical ventilation with supply air is it mainly two types of supply types that are used; the air inlet is close to the ceiling or displacement ventilation with large supply ducts placed at the floor level with a low air velocity [45]. Natural ventilation is not used in new houses in Sweden based on the cold climate and the fact that infiltration rates should be low.

Thermal storage is a parameter that is affected by the heat capacity and density of the different building materials and the heat losses from the building. The increased insulation and overall thicker walls in new buildings cause the thermal inertia to increase. This affects the heat demand of the building, since the maximum load is decreased and an offset in heat losses is created compared to the outdoor temperature.

2.1.3 Outdoor conditions
Outdoor conditions affect the heat transfer through the building elements and also the heat demand for ventilation and heat to cover infiltration. The parameters that have the largest impact are the outdoor temperature and solar radiation. Wind speed affects both the external convective heat transfer coefficient and the infiltration rate.

2.1.4 Internal heat
Internal heat is the largest uncertainty for predicting the heat demand for a building, especially the part based on the human behavior. The human behavior affects the supply heat from the equipment and appliance load, occupant load and lighting. Intended and future user behavior should therefore be assessed carefully [46]. Neto and Fiorelli [47] suggest using +20% of the building energy usage for the internal load when performing sensitivity analysis.

2.1.4.1 Heating and ventilation systems
Heat distribution is mainly supplied by three types of systems, hydronic, air or direct systems. In hydronic systems the most common supply methods are through radiators or with underfloor heating [53]. The heated water could also be used in order to heat the building through the ventilation airflow in a combined system. Direct systems are also common where electricity is used for heating electric radiators or an air/air heat pump, but also stoves not connected to a hydronic system are counted as a direct system. Air distribution systems are mainly used for ventilation but are often (always in cold climate) combined with a heater either from a direct system using electricity or by a heat coil supplied from the hydronic system.

The type of space heating units used for the different systems has a large impact on the indoor thermal comfort, since the space heating apparatus creates different temperature
Literature overview

gradients. The conventional radiator and underfloor heating system has shown to provide an overall good thermal climate [48]. The underfloor heating system with a high amount of radiation heat transfer exhibits no vertical temperature gradient [49]. Systems with local distribution (like air/air heat pumps and stoves) or air heating with ventilation supply air in only some of the rooms inside the building have been less studied from a thermal climate perspective. The thermal comfort was studied by Feist et al. [50] for a combined air heating and a pellet stove system.
2.2 Energy usages in buildings

The building energy usage is divided into three parts: embodied, operational and demolition. The embodied energy is the sum of all energy needed in order to produce the building and its associated systems. Additional embodied energy can be added to the building from renovation or extension. Operational energy is the space heating, hot water and the operating electricity (i.e. pumps and fans for heating and ventilation). The demolition phases include the energy for deconstruction and transportation of materials to landfill sites and/or recycling plants.

2.2.1 Operational energy

In section 2.1 the building heat balance is discussed; how this relates to the building’s energy usage is presented in this section. Operational energy is by definition the energy for maintaining the inside environment through processes such as heating and cooling, lighting and operating appliances [51]. The system boundaries as building energy usages can be defined as delivered energy, net delivered energy or primary energy usages. A schematic overview is presented in Figure 3. Where net delivered energy is bought in forms of district heating, electricity or fuels, if any energy is exported it is counted as a surplus. The delivered energy is the energy need including thermal conversion factors (like COP for heat pumps) and onsite renewables from i.e. solar cells, solar collectors or wind power but excluding sold energy from the building. The energy need in Figure 3 is related to building heat balance.

Different classifications for low energy buildings use different system boundaries. The Swedish building regulations use the delivered energy, while passive houses according to FEBY use the primary energy factor with (conversion factors 2.5 for electricity and 0.8 for district heating) for mixed systems and delivered energy in cases of a single supply system. For zero energy buildings the sold energy is also included inside the primary energy factor.
2.2.2 Embodied energy.
A review study by Sartori and Hestnes [52] showed that the operational energy is by far the largest consumer during a building’s lifetime for conventional and low energy buildings built before 2006. The embodied energy for newer buildings has been studied by Chastas et al. [53]. The study focuses on low energy buildings, passive houses and nearly zero energy buildings. The resulting review shows an amount of embodied energy versus operational energy usages over a 50-year lifespan. The study shows an embodied energy around 80% of the total energy usage, but these buildings were located in India, Lebanon and Brazil close to the equator. Buildings in Scandinavia, Sweden Norway and Finland have a higher amount of operational energy in relation to the embodied energy. It can be summarized such as more energy efficient buildings combined with the trend of on-site renewable energy will increase the shear part of embodied energy. The total amount of embodied energy in newly built houses has been slightly reduced with time. This is a consequence of material and system for the buildings having been more energy efficient to produce. [54].

2.2.3 Low energy buildings in Sweden
Low energy building has not a static definition. A building called low energy building twenty years ago will not be called a low energy building today [55]. Weber [56] has investigated results for 100 low energy houses in Sweden and Germany. The result shows an average overall UA-value at 105.34 W/K with a standard derivation of 57.37 W/K. This represents an average total specific loss factor of 1.21 W/m² K, where ventilation and infiltration losses were included. More recent studies of buildings like nine passive houses in Linköping studied by Rohdin et al. [57] had a designed space heating power of 20 W/m,
which represents a specific heat loss factor approximately of 0.55 W/m²K. The buildings had a higher space heating demand compared to the passive house standard with a 12 W/m² floor area at dimensioning outdoor temperature. The energy performance was around 21kWh/m².

On the whole to describe a building as a low energy house is quite unclear in Sweden. But other concepts like FEBY min energy house, passive house and zero energy house are clearly defined [58]. Other classifications are Sweden Green Building Council environmental buildings 3.0 that can be classified gold, silver or bronze and the international passive house standard. A summary of the requirements is presented in Table 2 for Kiruna as reference location, where the new FEBY standard with its gold, silver or bronze is defined. The new standard focuses on the heat loss factor and not on the energy usages; this is due to this requirement specifying the building performance and not being affected by the chosen energy system [59].

Studies of energy demand for passive houses in Sweden have earlier been performed. In Sweden the most famous example is the buildings in Lindås built in 2001, which are studied by Wall [60]. The twenty buildings show a space heating demand between 4.7 and 12.3 kWh/ m² yearly and a peak load at 5 to 8 W/m² floor area.

The concept of nearly zero energy buildings that will be implemented in the European Union’s states that all new buildings by 2020 should be nearly zero energy buildings and public buildings by 2018. For this implementation the Swedish building and planning has changed the system boundary to primary energy from a specific energy requirement [61].
Table 2. Comparison of different low energy concepts with the Swedish building regulations.

<table>
<thead>
<tr>
<th>Energy Usages</th>
<th>Heat loss factor</th>
<th>Indoor climate</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>FEBY (passive house)</strong></td>
<td>≤58, without electricity: ≤29 kWh/m² yearly</td>
<td>SVF &lt; 0.036 (^{(2)})</td>
</tr>
<tr>
<td><strong>FEBY Net zero energy building</strong></td>
<td>Weighted Sum of energy is equal to zero. Excluded lighting</td>
<td>SVF &lt; 0.036 (^{(2)})</td>
</tr>
<tr>
<td><strong>FEBY Mini energy</strong></td>
<td>≤78, without electricity: ≤37 kWh/m² yearly</td>
<td>SVF &lt; 0.036 (^{(2)})</td>
</tr>
<tr>
<td>Feby gold (2018) BBR</td>
<td>18 W/m² (higher value for buildings smaller than 600 m² floor area and a supply airflow larger than 0.45 l/s m²)</td>
<td>SVL &lt; 29 (^{(3)})</td>
</tr>
<tr>
<td>Feby silver (2018) BBR</td>
<td>21 W/m² (higher value for buildings smaller than 600 m² floor area and a supply airflow larger than 0.45 l/s m²)</td>
<td>SVL &lt; 29 (^{(3)})</td>
</tr>
<tr>
<td>Feby bronze (2018) BBR</td>
<td>24 W/m² (higher value for buildings smaller than 600 m² floor area and a supply airflow larger than 0.45 l/s m²)</td>
<td>SVL &lt; 29 (^{(3)})</td>
</tr>
<tr>
<td>Miljöbyggnad (Gold)</td>
<td>≤70% of BBR</td>
<td>Thermal indoor climate fulfills PPD ≤ 10% at DVUT.Comfort + inquiry or measurement</td>
</tr>
<tr>
<td>Miljöbyggnad (Silver)</td>
<td>≤80% of BBR</td>
<td>Thermal indoor climate fulfills PPD ≤ 10% at DVUT (^{(4)})</td>
</tr>
<tr>
<td>Miljöbyggnad (Bronze)</td>
<td>BBR</td>
<td>Thermal indoor climate fulfills PPD ≤ 15% at DVUT (^{(4)})</td>
</tr>
</tbody>
</table>
| BBR 22                         | ≤130, without electricity: ≤95 kWh/m² yearly                                      | Operative temperature \((T_{op}) > 18°C.
Maximum difference of 5°C among the single rooms.
Floor temperature in the range of 16 to 26°C,
Air velocity < 0.15 m/s during the heating period and < 0.25 m/s during the rest of the time. |
| BBR 25 (2017)                  | Primary energy sum <90 kWh/m²                                                     | BBR 22                                                                         |
| Passive house institute (Passive house int.) | Space heating 10 W/m² or Space heating 15 kWh/m²                                  | Above 25°C maximum 10% of the year                                             |

\(^{(1)}\) \(F_{geo}\) Constant based on location in Sweden (Kiruna 1.9)  
\(^{(2)}\) SVF = \(g \times \text{Aglas/Agolv}\)  
\(^{(3)}\) SVL = 800 \(g \times \text{Aglas/Agolv}\) Simulated values  
\(^{(4)}\) Simulated values
2.3 Thermal comfort

The most cited way to describe the indoor thermal comfort in buildings is according to Fanger’s comfort model [11], which is also used in the EN ISO 7730 [62] and the ASHRAE Standard [63]. The model calculates the number of dissatisfied people based on the temperature from convection and radiation, relative humidity, air velocity, metabolic rate and also clothing.

Thermal comfort is often investigated inside the occupied zone. The definition of the occupied zone in a building (the volume occupants normally use) is the volume enclosed by two horizontal planes, one 0.1 m above floor level and the other 2.0 m above floor level; vertical boundaries are 0.6 m distance from the exterior walls or 1.0 m from windows and doors [61].

The EN ISO 7730 [62] defines three categories of thermal indoor climate (A, B and C) presented in Table 3. The thermal comfort demand is based on Fanger’s comfort models: Predicted percentage of dissatisfied (PPD), the Predicted mean vote (PMV), the Draft ratio (DR) and the Local thermal discomfort (PD). The local thermal discomfort is divided into temperature gradient, floor temperature and radiant asymmetry.

Table 3. Thermal comfort according to EN ISO 7730.

<table>
<thead>
<tr>
<th>Category</th>
<th>PPD (%)</th>
<th>PMV</th>
<th>DR (%)</th>
<th>PD (%)</th>
<th>Vertical temperature difference (°C)</th>
<th>Range of floor temperature (°C)</th>
<th>Radiant temperature asymmetry (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>&lt;6</td>
<td>-0.2 to 0.2</td>
<td>&lt;10</td>
<td>&lt;2°C (PD &lt;3%)</td>
<td>19 to 29°C (PD &lt;10%)</td>
<td>PD &lt;5%</td>
<td></td>
</tr>
<tr>
<td>B</td>
<td>&lt;10</td>
<td>-0.5 to 0.5</td>
<td>&lt;20</td>
<td>&lt;3°C (PD &lt;5%)</td>
<td>19 to 29°C (PD &lt;10%)</td>
<td>PD &lt;5%</td>
<td></td>
</tr>
<tr>
<td>C</td>
<td>&lt;15</td>
<td>-0.7 to 0.7</td>
<td>&lt;30</td>
<td>&lt;4°C (PD &lt;10%)</td>
<td>17 to 31°C (PD &lt;15%)</td>
<td>PD &lt;10%</td>
<td></td>
</tr>
</tbody>
</table>

* Depend on 4 different temperature differences described in the EN ISO 7730 [62]

In the PPD and PMV models the factor that mainly affects the experienced indoor climate is the temperatures (both air temperature and radiation temperature). For indoor environments where the air velocities are small, usually around 0-0.15m/s in the occupied zone, the effects of air velocities are very small. The relative humidity has also a minor impact on the PPD and PMV values. The main parameter to describe correctly indoor climate, is the indoor air temperature and radiation temperatures on envelope surfaces. In Figure 4 below a psychometric chart describes the indoor climate according to the standard EN ISO 7730.

15
Figure 4. Psychometric chart there the area between the lines A, B and C represents a good thermal climate according to EN ISO 7730, a) winter clothing 1 clo and metabolic rate at 1 met, b) summer clothing 0.5 clo and metabolic rate at 1 met.

The operative temperature \( T_{op} \) is calculated according to Equation 1. Operative temperature is defined as uniform temperature of an imaginary black enclosure in which an occupant would exchange the same amount of heat by radiation and convection as in the actual non-uniform environment [62].

\[
T_{op} = \frac{h_c T_a + h_r T_r}{h_c + h_r}.
\]  

(1)

Where \( T_a \) is the ambient air temperature, \( T_r \) the radiation temperature, \( h_c \) the convection coefficient and \( h_r \) the radiation heat transfer coefficient. A simplified version is presented in Equation 2, which the air velocity \( v \) describing the relation between \( h_c \) and \( h_r \).

\[
\begin{cases} 
T_{op} = \frac{T_r + T_a \sqrt{10v}}{1 + \sqrt{10v}} & \text{if } v > 0.1 \text{ m/s} \\
T_{op} = \frac{T_r + T_a}{2} & \text{if } v \leq 0.1 \text{ m/s} 
\end{cases}
\]  

(2)

While PMV and PPD describe the comfort for the whole body, the DR and PD are local thermal discomfort parameters. The draft parameter most commonly causes local discomfort, but this can also appear for high temperature gradients, radiant temperatures asymmetry and with high or low floor temperatures. Draft ratio is a parameter based on air velocity, temperature and turbulence intensity stated by Fanger et al.[64]. The model is used to quantify draft risk and when designing air distribution systems with a low risk of draft. Problems with high discomfort based on draft occur mainly in air-conditioned buildings [64]. The draft ratio is presented in Equation 3, where \( I_u \) is the turbulent intensity.

\[
T_{op} = \frac{T_r + T_a \sqrt{10v}}{1 + \sqrt{10v}} \text{ if } v > 0.1 \text{ m/s} \\
T_{op} = \frac{T_r + T_a}{2} \text{ if } v \leq 0.1 \text{ m/s}
\]
\[ DR = (34 - T_a)(v - 0.05)^{0.62}(0.37v I_u + 3.14) \quad (\%) \quad (3) \]

The building rules in Sweden state that buildings should be designed so that a satisfying indoor climate is obtained. In order to achieving a tolerable indoor thermal climate, the following criteria should be fulfilled within the occupied zone of the building [61].

- Operative temperature \( (T_{op}) \) not lower than 18°C.
- Maximum difference of 5°C among the single rooms.
- Floor temperature in the range of 16 to 26°C.
- Air velocity not higher than 0.15 m/s during the heating period and 0.25 m/s during the rest of the time.

These criteria can be omitted if the constructor proves that a satisfying indoor climate will be achieved anyway.
2.4 Simulation methods

Indoor climate simulations are an important tool for designing the heating system, comparing different construction solutions and evaluating the building performance. For low energy buildings the importance of using simulations has increased. Most of the different classifications criteria of the building require certain ranges of energy performance and indoor climate. Simulations of energy usage and the indoor climate in buildings are mainly performed with three different methods. The simplest method is the multi-zone method and the most advanced method is the CFD approach. The most used method is the Multi-zone model that is used by the commercial software like TrnSys [8], EnergyPlus [7], IDA-ICE [6] and ESP-r [65]. The zonal method is a compromise between CFD and the multi-zone model.

2.4.1 Multi-zone Method

The multi-zone method is the most popular simulation approach for investigation of buildings, used for all types of low energy and environmental classifications. The most common way is to solve the energy balance inside the building with solving transfer functions or by finite difference. The software uses a 1D method where the thermal transfer equations are solved in each node of the system (see Figure 5 a). The main advantages of the method are that the heat balance can be solved with a large time scale and short simulation times. The method is especially adapted for energy usages [5]; this method accounts for the thermal storage inside the building, which is not possible for manual calculations.

The airflow is calculated based on a network model where the airflow into the zones and between the zones is calculated with a one-dimensional expression. The method uses a well-mixed assumption for the flow with constant temperature in each zone. This makes it impossible to investigate local temperature gradients inside the rooms [66]. The airflow between different zones through openings is based on a pressure difference equation (calculated with simplified Bernoulli equations) and the mass flows is often calculated from a simplified (steady isothermal) orifice equation including a discharge coefficient [67]. In multi-zone software the pressure difference is commonly based on a hydrostatic pressure assumption. This method will mostly properly produce significant errors for the airflow [68]. For multi-zone airflow simulations a large uncertainty is also the definition of the discharge coefficient and the behavior of horizontal openings in thermally driven flows [69]. This makes it difficult to simulate the indoor climate for local distributed heat systems where the system relies on a heat transport between the rooms with air. For systems with heat distribution in every single room this will not be a large problem.

Another drawback is the 1D approach used by the software packages; thermal bridges must be added to the heat losses manually and as described in section 2.1.1 the errors in the standard methods are large.
2.4.2 Computational Fluid Dynamics

CFD is the most advanced of the building simulation methods, based on the Naiver Stokes equations in order to solve the flow field in the fluid domain of the building [5]. The building volume is discretized in small cells (see Figure 5c), which makes this method computationally demanding to solve [70] and requires small time steps and a fine grid [71]. This makes the method not suitable for dynamic problems for energy usages and thermal performance of a building, since the solid walls have large thermal inertia, which affects the heat transfer in the building. The method is mostly applicable for indoor climate simulations for steady state cases, different heating/ventilation/distribution systems or when specific details in the construction are studied. The method presents differences in the thermal climate and indoor air quality parameters in detail on local levels for the whole simulation volume [5]. The method can also describe all parameters that affect the thermal comfort like air temperature, radiation temperature, air velocity, relative humidity and also the turbulent intensity on a local scale.

2.4.3 Zonal Method

The Zonal Method is a mix between CFD and the multi-zone model. The simulation volume is discretized to cells like in the CFD model but larger cells are often used. The difference is that the turbulence model equations and the naiver-stokes equations are not solved. The zones are connected by interfaces mass flow and heat flux is transported between each cell [72]. The method solves the pressure field to predict the airflows and temperatures inside the cells. The mass flow between cells is often calculated from the pressure difference based on a power law formulation [73] but other couplings between pressure and airflow also exist. On the whole it is a simplified version of CFD in order to solve the indoor airflow, mass and heat transport with reduced simulation time and the ability to have larger time steps. A drawback is that this approach needs complementary information and models to define flows [70]. Studies with zonal models compared to CFD have in some cases showed good agreement, which is especially true for mechanically driven flows, while larger variations to CFD and measurements have appeared for naturally driven flows, which is probably due to lack of accuracy in the physical representation of free convection [74]. Mora’s et al.[72] studies have compared CFD and zonal modeling. This study shows that the temperature and velocity field inside the building can be solved with the zonal method. But course grid CFD is preferred due to the better accuracy, since the simulation times with course grid CFD is still short. According to review paper by Chen [75] zonal models have still a long way to go to be a user-friendly and reliable tool for ventilation designers and the method will properly be replaced by coarse-grid CFD simulations in the future based on the small difference in computational demand between the models.
2.4.4 Coupled multi-zone and CFD simulations

Work is also performed with coupled CFD and multi-zone analysis in order to improve the airflow studies compared to the stand-alone multi-zone modeling. The idea behind the coupling approach is that the multi-zone gives pressures or flow rates to CFD and the CFD simulation solves and returns the other parameter to the multi-zone simulation in order to predict the airflow between rooms better. This is used for rooms where the well mixed assumption fails [76]. A schematic Figure for a coupled CFD and multi-zone example is presented in Figure 5b. For natural convection a coupled model has shown better results compared to the multi-zone stand-alone and the result is reasonable compared to the CFD model [77]. Wang and Chen [76] studied both airflows in an office suite and natural cross flow ventilation for a four zone building. The convergence between the different coupling methods was studied with measurements. The results show that the coupling method was not as accurate as the CFD model for all the zones but still an improvement of the multi-zone model.

![Figure 5](image)

**Figure 5.** Schematic outline of the discretization of a) multi-zone, b) coupled multi-zone and CFD, and c) CFD.

2.5 CFD in buildings

The first studies that used CFD to predict indoor airflow was made in 1974 by Nielsen [78]. With improved computer performance and software applications CFD studies of thermal indoor climate have steadily increased in the last decade.

The work has been focused on single rooms. Nielsen *et al.* [79] have summarized the progress in the field of room air distribution. The paper concluded that with faster computers, special situations can be investigated with CFD simulations like several interconnected rooms in a building and the connection between air distribution and heat consumption. For indoor airflow Zhai *et al.* [80] have summarized different turbulence models. The ordinary **k-ε** models have shown acceptable results for main flow field and temperature gradients with good computational economy. The model has however difficulties with high buoyant
flow (large part natural convention) and large temperature gradients. More advanced turbulence models like v2f (volume to fluid) have shown to be promising and LES have shown more detailed results. These models are though far more demanding in simulation time and computer performance than the k-ε model.

Thermal indoor climate with different heating systems and ventilation systems has been studied with CFD. Myhren and Holmberg [48] investigated a radiator system and an underfloor heating system for an office located in Sweden during winter conditions. A four-way air supply device, installed for cooling purposes was studied for a classroom with CFD; the investigation was performed by Noh et al. [81]. Gilani et al. [82] investigated a room with displacement ventilation combined with a heat source.

CFD with mixing ventilation was studied by Lee and Awbi [83]. They investigated the airflow for an open-space planning room and validated the results with smoke visualization. Lin et al. [84] have compared mixing ventilation with displacement ventilation. They showed that the comfort level according to draft ratio is similar if displacement devices have a supply velocity below 0.2 m/s. Chiang et al. [85] have studied mechanical ventilation with cold ceilings in a subtropical region. The cooling capacity was improved when the supply air was moved to the floor level.

CFD and natural ventilation are well studied both for wind and buoyancy driven flows. The wind driven flow has been studied and validated by Evola and Popov [86], for both cross-flow and single-sided natural ventilation. Both the k-ε standard model and the RNG turbulence model were compared, and both models showed good agreement with experimental data. For the field of wind driven natural ventilation LES models have also been studied, for example by Jiang and Chen et al. [87], who simulated natural ventilation with a Smagorinsky subgrid-scale (SS) model and a Filtered dynamic subgrid-scale (FDS) model. Buoyancy driven natural ventilation has been studied by Jiang and Chen [88]; a single-sided natural ventilation with large openings was simulated with both analytical, RANS and LES models. The work concluded that the LES models provide the best results, but based on computer time a RANS model was the obvious choice of turbulence. The analytical approach resulted in a good estimation of the flow if the discharge coefficient was set correctly. For buoyancy driven natural ventilation in large buildings the total domain was studied. Liu et al. [89] simulated and validated a building with an atrium using the RANS turbulence model. The temperature difference was maximum ±2K between simulated and measured data. Overall the natural ventilation method is not commonly used for buildings in a sub-arctic climate.

The CFD method has frequently been used for describing the indoor air quality (IAQ). A higher flow rate decreases the contamination inside a room. These CFD studies were performed both in order to keep sufficient ventilation flow rates, where air change efficiency
is used as parameter [90], and also specific pollutions transports like CO$_2$, Radon or asbestos. There CO$_2$ is the most common studied pollutant, for example by Wang et al. [91] that studied CO$_2$ concentrations in a classroom.

Micro climate (close area around a human body) can be studied in different ways with CFD. The thermal climate can be investigated as skin temperatures or IAQ when using different heating and ventilation systems for a building. Gao and Niu [92] have reviewed the field for thermal environment, and they concluded that the geometry of human beings has influence on the local airflow close to the body but not on the global airflow.

Thermal comfort with CFD simulation for a whole building or multiple rooms is less studied. A full-scale building is mostly used for fire safety simulations and natural ventilation studies. Hussain et al. [93] simulate the thermal comfort for a simple three-story atrium building with buoyancy driven natural ventilation, where PMV adjusted for natural ventilation was used for evaluation of the thermal comfort. Muhsin et al. [94] simulated a multi-story building with wind driven natural ventilation and validated the velocity at five different locations. Fire and smoke behavior is also studied, where Kolaitis et al. [95] studied gypsum plasterboard walls, with CFD for a full scale two-story building and Hostikka et al. [96] studied fire induced pressure and smoke spread for a nearly zero energy building.

2.5.1 Low energy buildings and Passive house studies.

Rodin et al. [57] investigated energy demand and indoor climate in a passive house located in the south of Sweden using multi-zone modeling with IDA ICE, CFD simulations and a post-occupancy evaluation. The system used for the simulation was an air heating system with displacement devices and a maximum heating supply of 2 kW. The local temperatures from the model were validated with a thermal imaging for a supply room on a paper sheet. The work also studied the PPD and PMV values inside the building using CFD results.

Karlson and Moshfegh [97] simulated a passive house in Lindås, Sweden with CFD. The building had an air heating system with displacement devices for a summer, winter and autumn case. Calculated heat fluxes from U-values were set as boundary conditions for the envelope surfaces and airflow values for supply air from measurements. Air velocities and temperatures were studied.

Feist et al. [50] studied if a good thermal indoor climate was achievable according to EN ISO 7730:2005 for a test room with an air heating system. A case study for combined air heating and a stove was also simulated with a multi-zone model and a CFD model was used for verification of the multi zone model.

Bajc et al. [98], investigated effects of Trombe walls; the energy demand for moderate continental climate was investigated with a CFD model for one room. Fokaides et al. [99] investigated the indoor climate for a passive house in subtropical climate. A multi-zone software energy plus was used in order to predict the climate, and a more detailed study
using CFD for one of the rooms was performed. Wang et al. [100] have investigated a classroom with CFD for a passive school building using displacement ventilation.

2.6 Summary of literature and motivation

In the field of energy and indoor climate simulations for buildings the multi-zone model is the mainly used method today, but CFD is a strongly increasing method especially in research work. The building design and construction industry has not so far adopted the CFD method mainly due to large simulation times and costly licenses [101].

The multi-zone method is based on a 1D approach for solving the heat balance; this causes uncertainties due to thermal bridges and heat losses through the foundation of the building. The main advantage of the method is a fast simulation time and still reasonable results in the form of the energy performance of the building. The well-mixed assumption used for the air with no temperature gradients makes indoor climate simulations lacking in accuracy. This is mainly a problem due to the low-energy building trend with small heat requirements and no traditional heating systems with heat supplied in each room of the building.

CFD simulations are in contrast to the multi-zone approach much more computation demanding and the simulation times are too long in order be a useful tool in the design process of the building. Even though CFD is superior, the multi-zone approach in prediction of the indoor climate is not commercially used to a large extent. In order for the construction industry to be able to use CFD simulations more frequently, course grid simulations must be performed. The turbulence model for course grid simulations has problems when predicting the natural and mixed convective flow. This is due to the wall function being adapted to high Reynolds number flow, which does not appear in most buildings, except close to the ventilation inlets. Another drawback of the method is that the fluid flows require small time steps. Transient simulations therefore require a longer simulation time.

Based on the long simulation times and standard wall functions lacking in accuracy for natural and mixed convection, the performance of locally distributed air heating systems for many rooms or whole buildings is less investigated using CFD.

More studies concerning both supply and exhaust air to different rooms is needed for an air heating system; validation of simulated data should be performed. Studies concerning multi-story buildings should be implemented in order to properly design air-heating systems according to buoyancy flow heat transport.

Building energy performance simulations in a sub-arctic climate differs from more southern locations, since the colder temperatures make a snow covered landscape for a large part of the year. This affects the thermal load of the building because of the insulation properties and thermal inertia of the snow. It is mainly the foundation heat losses but also the ceiling
losses that will be affected. The snow-covered ground will have a higher soil temperature compared to an uncovered surface.

Multi-zone models used for foundation heat losses do not include the effect of snow. The 3D simulation models used for predicting soil heat transfer problems have mainly not included snow and the earlier studies were performed in climate conditions with small amounts of snow and during a short period. The model by Rees et al. [29] has constant properties of snow and a snow depth of maximum around 0.2m. More studies of how snow affects the foundation heat losses in more snow rich areas are of importance.
3 Studied systems

In this work three different buildings were studied (building 1-3), all buildings have lower energy demand than a standard single-family house and are located in a sub-arctic climate. Figure 6 shows the floor plan and geometry of the buildings and Table 4 presents U-values for the envelope surfaces of the buildings and investigated heating systems. Building 1 is studied in paper I, building 2 was used for comparison of different heating systems in paper III and building 3 was studied in paper IV in order to evaluate an air heating system in a sub-arctic climate. In paper V four common foundation types in Sweden were studied in order to evaluate if snow cover is an important parameter for building simulations. The different foundations are presented in Figure 7.

3.1 Studied Buildings

Table 4. U values and heating system for the studied buildings.

<table>
<thead>
<tr>
<th>Building</th>
<th>U-Values [W/m²K]</th>
<th>Heating system</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Walls Ceiling Floor Window Door</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>0.109 0.085 0.150 0.85 1.3</td>
<td>Underfloor heating</td>
</tr>
<tr>
<td>2</td>
<td>0.109 0.085 0.150 0.85 0.9</td>
<td>Underfloor heating, Air/air heat pump, air heating</td>
</tr>
<tr>
<td>3</td>
<td>0.079 0.039 0.065 0.65 0.7</td>
<td>Air heating</td>
</tr>
</tbody>
</table>

Building 1

Building 1 is a one-story house; all rooms have a ceiling height of 2.4 m, except the combined kitchen and living room. The floor area is 109 m² with a floor plan and geometry according to Figure 6a. The building is located in Blåsmark outside the town of Piteå. The house has five ventilation supply devices placed in the bedrooms and in the living room. The supply air flows through a channel that is placed in the soil before it enters the building. The total supply mass flow rate is 0.04 kg/s. The exhaust air devices are located in bathrooms and kitchen. The building is heated with an underfloor heating system.
**Studied systems**

**Building 2**

Building 2 was a projected 98m² one floor single-family house with 3 bedrooms and with a ceiling height of 2.4 m located in Kiruna. The floor plan and building geometry are presented in Figure 6b. For this building, three different heat distribution systems were studied: an underfloor heating system, an air heating system, and an air/air heat pump system.

- The underfloor heating system was installed in each room of the house.
- The air heating system used displacement devices for the supply air; they were located in the bedrooms and the living room. The supply air temperature was 45°C with a total supply flow rate of 66 l/s for the studied case, in order to fulfill the heat demand with an outdoor temperature of -30°C.
- The air-to-air heat pump was installed above the entrance door and operates as a local heat source. The airflow rate through the condenser unit was set at 0.164 kg/s and a supply temperature of 35°C.

The ventilation air supply flow rate was 34 l/s for the underfloor heating system and air/air heat pump case.

**Building 3**

Building 3 was designed according to FEBY’s passive house standard and is located in Kiruna. The building is a semi-detached house with a floor area of 140 m² for each apartment in two stories, where every apartment has a 14 m² solar cell installed. The building was constructed with an air heating system, with a rotational FTX system. The supply air heating coil is heated by district heating connected to a low temperature district heating system. The primary system has a maximum supply temperature of 70°C. Each apartment had separately installed systems. The showers used in the building recirculate the water and reheat it with electricity.

The building was studied with CFD at two different outdoor temperature conditions -5°C (validation case of 14:th December 2016) and the design outdoor temperature -25.7°C.
Figure 6. Floor plan and geometry for the different studied buildings, a) building 1, b) building 2 and c) building 3. (In figure B = Bedroom, Bath = Bathroom, L = Living room, K = Kitchen, La = Laundry room and HP = heat pump location, 1-6 is measurement locations for Building 1, P1-P4 for building 3 and L1 is location of vertical line for comparison of different heating system).
3.2 Foundations

There are three types of foundation constructions that dominate the market in Sweden for single-family houses. The different foundation types are crawl space, slab and basement foundations. Each type has around 25-35% of the market [102] and the total number of single-family houses is around two million today [103]. Slab foundation is the most common option in newly built single-family houses, while crawl space foundations are mainly used for modular housing when the buildings have already been constructed with a joist floor. The method is used by many of the largest constructors of single-family houses in Sweden. The basement type was more common several decades ago.

The cross-sections of the different foundation types studied in this work are presented in Figure 3. The different types examined were an uninsulated crawl space foundation (A), an insulated crawl space foundation (B), an insulated slab foundation (C) and a slab foundation with extra insulation (D) against the vertical edge of the foundation in order to break thermal bridges.

All the compared foundation types were simulated using the same internal floor area. The foundation of the building had an inner dimension of L*W: 11.26x7.06 m. The dimensions L1 to L4 and H1 to H9 are presented in Paper V.
Figure 7. Different foundation configurations: a) uninsulated crawl space, b) insulated crawl space, c) insulated slab, d) slab with extra insulation.
Studied systems
4 Validation procedure

For CFD simulations validation is of importance to investigate if the model represents the physical reality. In this work Building 1 and 3 was validated for indoor temperature in order to investigate if the indoor climate can be predicted with CFD. The heat losses from a foundation in Paper V was also validated against measured temperatures. In this section the validation procedures are described.

4.1 Validation of Building 1

The CFD model of building 1 was validated against temperatures at 6 different locations in Paper I. The location of the sensors is presented in Figure 6a, at a height of 2 m above the floor level. The temperature sensors were of type pt1000, with an accuracy of ±0.4°C. The supply air was assumed according to Boverket:s minimum flow regulations at 0.35l/s m² floor area [61], with the flow distributed according to BBR 10 [104], presented in Table 5. The day chosen for validation was 2014-03-13, at the time 6:00 based on that the lowest variation in outdoor temperature for the measurement period appear that morning and that the impact of solar radiation is small furthermore the location of the occupants is also known. The measurement period was between 2014-03-01 to 2014-04-30, the supply heat to the under floor heating system was measured (1.2kWh during the time for the validation) and the supply air temperature was measured. Because lack of information of the heating system the heat was distributed even in the whole building, and heat from occupants was added according to Sveby [105] with 60W for the children in bedrooms and 100W per adult person (2 persons) in the large bedroom (as an additional heat flux for the floor heating system). Heat from equipment was not consider in the model. The transmission losses were calculated from overall heat transfer coefficients in Table 4, the outdoor temperature and area weighted indoor temperature was taken from measurements.

<table>
<thead>
<tr>
<th>Room</th>
<th>Supply mass flow (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Living room</td>
<td>0.013</td>
</tr>
<tr>
<td>Bedrooms</td>
<td>0.005</td>
</tr>
<tr>
<td></td>
<td>(for each room)</td>
</tr>
<tr>
<td>Large Bedroom</td>
<td>0.010</td>
</tr>
</tbody>
</table>
4.2 Validation of Building 3

The validation of Building 1 was based on measurements, calculations and some assumptions. To get a more detailed validation of the CFD model, extensive measurements were carried out in Building 3. For thermal comfort the temperature gradient at four different vertical lines from floor to ceiling was manually measured with a height above the floor of 0.1 m, 0.5 m, 1 m, 1.5 m, 2 m and 2.45 m. The measurements were performed with a temperature sensor on the 14th of December 2016; the installed sensors were calibrated for an error less than ±0.5°C. The lines were chosen to be representative for the whole building. They were located at both stage levels and rooms with both supply air and exhaust air. The lines were located inside the occupied zone and two of them were chosen around 1 m from a window. The air velocities were measured with a hot wire anemometer with a measurement error at ±0.03 m/s and ±1% of the measurement value.

To improve the boundary and initial condition setup from the simulation of Building 1, Building 3 was equipped with more measurements equipment for each apartment. In total 23 sensors were installed where 10 measures temperature and relative humidity inside the building. 5 sensors measure the temperature for the ventilation heat exchanger and heating coil. District heating was measured for both space heating and hot water. Electricity meters were installed for the total electricity to the building, the supply to the car engine heater, solar panels and each of the showers. From the measured heat rate to the ventilation heating coil and the temperatures before/after, the supply flow of the ventilation system was estimated, the ventilation system is balanced why the exhausted airflow was assumed to be the same as the supply airflow. For every specific supply device, the airflow was distributed from velocity measurements. The supplied airflow rates are presented in Table 6, exhausted airflow devices are located in bathrooms on each floor and the laundry room. The supply temperatures are presented in Table 6 where the Case 1 represents the validation temperatures based on measurements and the Case 2 represents the simulation data at design outdoor temperature.

The heat losses for the ventilation system was calculated from heat exchanger temperature efficiency that was measured and assumed equal to the heat exchanger efficiency when no humidity is added to the air and no occupants was living in the building. The infiltration losses were decided from the 50 Pa pressure difference measurements (87 l/s) and calculated with the EN ISO 13789 standard [106] and a shielding coefficient of 0.1. The transmission losses were calculated with U values from the manufacturer, area weighted average indoor temperature and a modified outdoor temperature described in section 4.3 based on metrological data for Kiruna for the measurement period. Heat losses calculation were carried out during the period 27 November 2016 and 15 January 2017 and compared with the measured heat supply during this period.
Table 6. Supplied air data for building 3.

<table>
<thead>
<tr>
<th>Room</th>
<th>Mass flow (kg/s)</th>
<th>Temperature (°C) Case 1 (Case 2)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Living room (ground floor), 2 devices</td>
<td>0.0150</td>
<td>34.7 (45.4)</td>
</tr>
<tr>
<td>Living room (first floor)</td>
<td>0.0085</td>
<td>34.7 (45.4)</td>
</tr>
<tr>
<td>Bedrooms (first floor)</td>
<td>0.0125</td>
<td>34.7 (45.4)</td>
</tr>
<tr>
<td></td>
<td>(for each room)</td>
<td></td>
</tr>
<tr>
<td>Bedroom (ground floor)</td>
<td>0.0140</td>
<td>34.7 (45.4)</td>
</tr>
</tbody>
</table>

4.3 Validation of Crawlspace

The thermal model for comparing different foundations is validated for a crawlspace foundation of type A in Figure 7a. The measurements were performed with 15 min sampling interval with eight calibrated temperature sensors inside the crawlspace. The crawlspace has a depth of 0.75 m and the sensors were located at a distance of 10 cm beneath the joist. Total 8 sensors were used, see Figure 8 for location. An average temperature inside the crawlspace was calculated from the measurements’ values. Inside the crawlspace a dehumidifier was located; the electricity was measured to the unit and calculated based on average values on a daily basis. Snow depth and outdoor temperature for the model were taken from metrological data.

![Figure 8. Sensor location for the crawlspace.](image-url)
Validation procedure
5 Numerical methods

The main part of this thesis concerns computational fluid dynamics of thermal indoor climate. This section summarizes some of the relevant theory in order to develop a CFD model for thermal comfort simulations in sub-arctic climate, including important parameters and possible simplifications for the CFD simulation. The simulation software used for CFD simulations in all five papers was Ansys CFX. More detailed descriptions of the methods are presented in each of the papers.

5.1 Grid refinement

To minimize the number of cells in the calculation, a discretization error study was performed in Paper I in order to find a maximum grid size that still does not heavily influence the results. The flow field and temperatures are the studied parameter, since these are the most interesting variables for the thermal comfort. Discretization error is numerical errors caused by the CFD model dividing the volume into a finite number of cells. The errors cannot be totally avoided, but it is of importance to know the size of the errors. The method used for comparison between different grid sizes was Roach grid convergence index (GCI) together with an extrapolated curve according to Richardson’s extrapolation [107]. Richardson’s extrapolation value represents an infinitely fine mesh.

In order to study the discretization error a two-room model was used. The total floor area was 16m² with a radiator in each of the rooms; one room had supply air and the other exhaust airflow.

The mesh was refined in order to have a constant refinement rate. The refinements were made by reducing the maximum size of the elements. This gave three grid sizes that were used. The number of elements for the simulation was 111k elements, 334k and 883k elements. For a grid convergence study it is important that the results must converge against a single value. An additional refinement was made with 2400k cells in order to investigate if the grid refinement converges against the extrapolated value.

5.2 Turbulence models and wall functions

In CFD simulations the turbulent flow motions are mostly solved with a turbulence model. The most used turbulence models are the k-ε models. The models are a two equation RANS (Reynolds average Navier Stokes) models that predict the turbulent kinetic energy (k) and the eddy dissipation (ε). Other commonly used models are the k-ω models, which also are a RANS model with the ω-formulation instead of an epsilon formulation; the ω is the specific dissipation. The SST model combines the two models in order to fit a wider range of
Numerical methods

applications. The SST model uses a k-ω-formulation in the near wall region and k-ε for the rest of the flow [108].

Turbulence models use wall functions to solve the flow and heat transfer inside the boundary layer, where the k-ε models most commonly use scalable wall function and the k-ω models use an automatic wall function. The scalable wall functions use a linear formulation in the viscous sub-layer if the y+ value is less than 11.06 and the standard logarithmic law is used for the outer layer if y+ ≥11.06. The automatic wall function switches instead to a low re formulation at y+<2; above y+ 30 the standard wall function is used and between these values a blending function is used [109].

The y+ value is a non-dimensional distance from the wall and used to describe the grid size in relation to the specific flow pattern. For indoor airflow simulation, the number of cells is often high due to the large volume of the building.

The scalable wall function is adapted for forced convection flow, but has problems with predicting natural convection heat transfer and the low Reynolds formulation for the automatic wall treatment requires a y+ value around 1 to perform well for natural convection [109]. This requires a grid size that is too fine in order to be practical for indoor thermal comfort simulations. In order to simulate the heat transfer with acceptable numbers of cells a user defined wall function (UDF) was implemented. The UDF is based on a theoretical formula for the convective heat transfer coefficients, where the term u*/T+ in Equation 3 is replaced with a normalized wall heat flux coefficient (w). The wall heat flux coefficient is calculated according to Equation 4, where the heat transfer coefficient (h_c) is included together with two coefficients C1 and C2. The C1 is a scale factor in order to simplify the geometry; the constant is set at 1 if the real geometry is modeled. The constant C2 is based on the simulation software using the near wall temperature (T_f) and the theoretical formula for h_c uses a bulk temperature in the fluid (T_ref). The C2 formula is presented in Equation 5.

\[ q_w = \frac{u^*}{T^+} \rho C_p (T_w - T_f) \]  \hspace{1cm} (3)

\[ w = C_1 C_2 \frac{h_c}{C_p \rho} \]  \hspace{1cm} (4)

\[ C_2 = \frac{(T_w - T_{ref})}{(T_w - T_f)} \]  \hspace{1cm} (5)

This method was implemented for a radiator in Paper II, where T_{ref} was located 1 m ahead of the radiator and 1 m above the floor level according to the reference point in the ISO EN 442-2 standard [110]. The UDF was also included in paper IV where heat transfer coefficients according to Awbi and Hatton [111] were used for all building surfaces.
5.3 Boundary conditions

For CFD the fluid flow time steps need to be short in order to capture the flow field. This makes simulations with the building envelope included in the model suffer with long simulation times. The advantage of including building envelopes is mainly transient behaviors where the thermal inertia of the building will be included in the simulation. This is especially important for validation studies where a steady state behavior of the heat losses is almost impossible to achieve based on fluctuations in outdoor temperatures. This is even harder to achieve in low energy buildings that are more insulated and also have a higher thermal mass.

An approach to including the transient behavior for validation of the thermal comfort is to calculate the heat losses by using a backward weighted average temperature. This can only be done if the amount of heat to the building is measured or can be estimated from the building simulation software. The backward weighted average outdoors temperature was calculated according to Equation 6, where the number of days used was selected from measurements of the heat losses.

\[
T_w = \frac{\sum_{i=0}^{n-1} T_i}{\sum_{i=0}^{n-1}}
\]

This method was used in paper IV together with the building U-values and infiltration rates for the building in order to calculate the heat losses. With this approach a steady state simulation of indoor environment can be used without including the building envelope in the CFD domain.

5.4 Soil and snow modeling

Different foundation types were studied in Paper V in order to investigate the effects of snow cover on the heat losses. For the foundation of a building heat transfer is mainly caused by conduction. Therefore, the fluid flow of the air inside the foundation (crawl space type) was assumed in the model to be well mixed, and no flow equations were solved in the simulations, but radiation was included. The convective heat transfer coefficient was treated with user-defined wall functions.

The developed model was a transient simulation model with 24-hour’s time step. To get an initial guess the model had a first run for a whole year to achieve an approximate distribution of temperatures in the soil around the foundation due to its high thermal inertia. The environmental conditions in the model for snow and air temperature were retrieved from metrological data. The boundary conditions and geometry were set according to Figure 8.
In a cold climate the soil temperature is affected by the thermal insulation from the snow layer and soil freezing. The thermal conductivity of the snow is based on the density. In the simulation model the thermal conductivity is established on a polynomial fit to measurements performed by Keller et al. [112]. The polynomial expression is presented in Equation 7, where the density is calculated based on the water equivalent and average values for the snow depth from weather station data. Assumption was made that no mass is removed from the snow layer before the density reach 550 kg/m$^3$ and the density is constant in the whole snow layer. In reality the snow density will vary through the different layers of the snow, and melting and freezing will occur. The specific heat capacity used for the snow was set to 2.08 kJ/kg K with a peak added for the range between -0.15 to 0°C, in order to include the latent heat of the snow and prevent the snow layer to reach above 0°C.

\[
\lambda_{\text{snow}} = 4 \cdot 10^{-6} \cdot \rho_{\text{snow}}^2 - 0.0013 \cdot \rho_{\text{snow}} + 0.2122
\]

(7)

For the ground freezing in the model, heat capacity and thermal conductivity are calculated according to Johansen’s method [113]. The soil properties used in the model were soil water content of 0.09 m$^3$/m$^3$, porosity of 0.4 m$^3$/m$^3$ and dry density of 1600 kg/m$^3$, which are the same conditions as those reported in Pericault et al. [114]. The heat capacity and thermal conductivity used are based on the temperature presented in Figure 9.
Figure 9. Soil properties versus temperature.
6 Results and Discussion

This section presents the most significant results from the different papers related to the objectives presented in the introduction. More detailed results may be found in the different papers.

6.1 Grid convergence index

In order to investigate how the indoor parameters’ temperature and velocity are affected by different grid sizes, a grid independency study was performed in Paper I. The study was of a two-room case according to the grid convergence index (GCI). In Figures 10 and 11 below the results of a grid convergence study are presented. The line used for comparison was located 1 m from the radiator in the exhaust air room; the exact location is described in paper I. Where Figure 10 represents the velocities and Figure 11 the temperature field for different grid sizes, the error bars in the right figures represent discretization error according to GCI. For the model, a second order discretization scheme and a Boussinesq approximation for buoyancy prediction were used. The turbulence model used in the simulation setup was the standard k-ε turbulence model.

![Figure 10](image-url). Velocity profile and discretisation error of velocities along studied line.
Figure 11. Temperature profile and discretisation error of Temperature along studied line.

The results in Figures 10-11 show that the discretization error is small on the whole in the simulation volume. The temperature gradient has also on the whole very small error bars; the results were almost identical for all the cases except the coarsest grid where some small deviations were seen.

The velocities have a maximum error around 20%, which represents an error of ±0.015 m/s. The highest discretization errors appear in a low velocity field 0-2m above the floor level (occupied zone for thermal climate) and are much smaller inside the buoyant jet close to the ceiling. From an indoor climate perspective, these errors are not so crucial since they appear in the regions with low velocity. The low velocity does not affect the thermal comfort in a sufficient way for Fanger’s comfort models PMV and PPD values. Temperatures have a higher impact on the thermal comfort according to these models.

Overall a 0.1m maximum face length for the model is sufficient in order to reduce the discretization errors. This grid size represents 313K cells for the two-room model. However, a courser grid can also probably be used without affecting the prediction of the thermal climate performance substantially based on the small deviations between the different grid sizes for the temperature profile.

6.2 User-defined wall functions

The result in paper I shows that the radiator surface and window surface had unrealistic temperatures. The radiator temperature had a too high surface temperature and the window surface temperature was too low. This is because the RANS turbulence models with scalable wall treatment have problems with predicting natural convection heat transfer. The model underestimates the natural convection heat transfer coefficients, which entails that a heat flux boundary conditions will give a higher/lower surface temperature and also give too high an amount of radiation heat flux. If instead a constant temperature boundary is
used, it will give a too low amount of convective heat transfer. This will mainly affect the surfaces with high heat flux or large temperature gradients like radiators and windows.

More advanced turbulence models using an automatic wall treatment have shown to be able to predict the natural convection heat transfer better. This is mainly due to the low Reynolds formulation being used in the model in the viscous sublayer for $y^+$ values below 11.06. For the automatic wall treatment, a $y^+$ value close to 1 is to be preferred to ensure the capture of the laminar sub-layer. A $y^+$ value around 1 requires a higher grid density that will increase the computational times significantly. In paper II, a method for treating natural convection heat transfer with user-defined wall functions is investigated for a radiator. This is used in order to be able to simulate the thermal indoor climate with better accuracy with a k-$\varepsilon$ model and does not increase the number of cells in the simulation volume when the wall functions are changed.

First a 2D model was investigated with a geometry presented in Figure 12a. For this model the standard k-$\varepsilon$ model with user-defined wall functions was compared with a standard k-$\varepsilon$ model with scalable wall functions and a SST model with automatic wall functions for a constant heat flux boundary. In Figure 12b the convective heat transfer coefficients for the different turbulence models are presented.

For the case with user-defined wall functions, the simulated heat transfer coefficient matches well with the coefficient from the SST model, while the k-$\varepsilon$ model with scalable wall functions heat transfer coefficients was significantly lower. The difference in the number of cells and the average $y^+$ values at the radiator surface for the different wall functions are presented in Table 7. The k-$\varepsilon$ model with user-defined wall functions has around 6 times fewer elements compared to the SST model. This will result in considerably lower simulation time for the model with the user-defined wall functions.
Results and Discussion

Figure 12. (a) Geometry of the 2D radiator, (b) Convective heat transfer coefficients for different turbulence models.

Table 7. Comparison of turbulence models considered.

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>Number of cells</th>
<th>Average y+ on radiator surface</th>
</tr>
</thead>
<tbody>
<tr>
<td>k-ε with scalable wall functions</td>
<td>3296</td>
<td>16.5</td>
</tr>
<tr>
<td>k-ω SST with automatic wall functions</td>
<td>27596</td>
<td>1.3</td>
</tr>
<tr>
<td>k-ε with UDF</td>
<td>4368</td>
<td>20.0</td>
</tr>
</tbody>
</table>

The k-ε model with UDF was also validated according to the Standard EN 442-2 test. Simulations with geometry of a real radiator model were performed; the geometry is presented in Figure 13a. The validation was made for three different water supply temperatures and three different mass flow rates for each temperature. In Figure 13b the results of the simulations are presented and compared with the Standard EN 442-2 test data. The simulated values agree with the predicted values.

For simulations of an entire building or a large room the geometry needs to be simplified so that the number of cells does not become unrealistically high, which will be the case if every detail is modeled. This is especially important for the area closest to a radiator, where
a real geometry with all the bends and convection plates on the radiator surface will lead to long simulation times and large memory requirements.

This part can be simplified by using a flat surface for the radiator geometry and add a constant to the user-defined wall function. The constant is to compensate for the increased heat transfer coefficient and give correct convective heat flux from the radiator without the need to model the real geometry of the radiator. This model is also compared with the EN 442-2 Standard and the result is presented in Figure 13c. The different cases are a flat radiator (Flat rad), an MP1 and an MP2 with a height of 300 mm. Also a case with a height of 590mm is presented. All the cases have a water supply/return temperature of 55/45°C and a width of 1m for the radiator surface.

The maximum difference is 1.4%, which appears for the radiator with 590mm height, but this is due to the constant used being adapted to the 300 mm radiator. For the 300 mm radiators the maximum difference was 0.2% for the MP2 radiator. The error is still small even if the constant is adapted to another height of the radiator. A higher surface temperature on a radiator will provide a higher amount of radiation heat flux from the radiator, which will maybe affect the thermal comfort according to Fanger’s models.
Results and Discussion

Figure 13. (a) Geometry of the model, (b) Comparison of CFD-simulation and Standard EN 442-2 for heat transfer rate vs water mass flow, (C) comparison simplified radiator model vs Standard EN 442-2.
6.3 Validation of thermal environment

Building 1

Simulation of Building 1 was validated at five points located in different rooms. The rooms where the temperature was measured were an open planned living room and kitchen, 3 bedrooms and one bathroom. The temperatures in the different rooms are presented in Figure 14 with horizontal location for the temperature sensors presented in Figure 6. The maximum difference between measured and simulated values was relatively small; the maximum difference was 0.75 °C.

![Figure 14. Validation of CFD simulation for building 1.](image)

Building 3

For building 3 a more extensive validation was performed. For this building, the heat losses from the studied object were recorded based on calculated values that were compared with the measured space heating and electricity for the building. The calculations were based on overall heat transfer coefficients using a backward weighted average outdoor temperature in order to include inertia to the calculations. The ventilation losses were based on measurements. The methods are explained in detail in paper IV. Comparisons between calculated and measured heat losses are presented in Figure 15a and the heat losses are divided into infiltration, ventilation and transmission during the validation period presented in Figure 15b. The calculated heat losses show good agreement with the measured value. The deviation was on average around 3%.
Results and Discussion

Figure 15. Calculated heating demand for building 3 compared with the measured one.

The comparison between measured and simulated temperature gradients is presented in Figure 16. The temperature gradients from the CFD model agree well with the measured temperatures. On average the difference between simulated and measured data was 0.2°C for the values inside the occupied zone. Outside the occupied zone larger deviations can be seen for the line P2. This room is a supply air room where the supply air temperature was fluctuating in periods of around 9 minutes between 21°C and 45°C. The fluctuation was based on incorrect control setting for the heating system. In the other supply rooms, the vertical line was chosen further away from the supply device, where the difference was on the same level as in the occupied zone.

For the simulated air velocities, the results show good agreement in 3 of the 5 measurement points. For the last 2 points the deviation was outside the tolerance of the measurement equipment. This is probably due to measurement values from the hot-wire anemometer being much lower than the calibration point at 1 m/s and the variations in supply air temperature making changes in the buoyancy flow field for the building. All measured velocities were lower than 0.15 m/s, which is a value where the thermal comfort is only slightly affected by the air velocities. The locations of the measurement points were all selected in areas where the velocity is normally higher than in the occupied zone.
The validation shows that building 3 has smaller temperature differences between measurement and simulation compared to building 1, although the heating system in building 3 is more complex. The better results are probably due to thermal inertia being considered and the fact that the measurements are more comprehensive for building 3.

**Figure 16.** Validation results for air temperatures: a) P1 and P3, b) P2 and P4.
6.4 Thermal comfort

The thermal comfort for three different heating systems was studied, a traditional underfloor heating system, an air/air heat pump and an air heating system, where the heat is distributed with ventilation air. The air system was investigated with displacement ducts located at floor level or mixing ventilation ducts at the ceiling. For the case with ducts at the ceiling, the building is a two-story building. All the cases that were compared were located in Kiruna and studied at the design outdoor winter temperature (-30°C for building 2 and -25.7°C for building 3); different DWUT values since the thermal time constant differs between the buildings. During this time the thermal indoor climate and the comfort requirements are hardest to fulfill. In the simulation models, all doors are open except for the bathroom door in building 2 and doors to bathrooms and laundry room for building 3. For building 2 the main parameter shown in the Figures was the operative temperature, since the different systems were compared with the Swedish requirements and for building 3 the PPD value was used based on the thermal climate being evaluated according to ISO 7730.

6.4.1 Underfloor heating system

The underfloor heating system studied for building 2 gave an overall good thermal comfort. The temperature difference over the entire house is small. The operative temperatures were around 20-21°C in the horizontal plane. This can be seen in Figure 17 a. The simulation results also show no vertical temperature gradients (see Figure 20). In Figure 17b the draft ratio is presented and it shows that the risk of discomfort from draft inside the building is less than 15% in the entire occupied zone; the air velocities are less than 0.15m/s in this area.

![Figure 17. Underfloor heating system for building 2, a) operative temperature distribution at 0.1 m, b) draft ratio at 1.1m above the floor level.](image)

6.4.2 Air heat pump

For the air/air heat pump case, the building was simulated with the unit located above the entrance door, see Figure 6. The operative temperature for this system shows the most uneven distribution among the considered systems for building 2. The vertical temperature
Gradients in the occupied zone are less than 2°C. For an air/air heat pump system the temperature gradients are mainly located in the horizontal plane. For this system, the internal air velocities exceed the upper limit of 0.15 m/s according to Swedish recommendations mainly in the hallways and in the living room. This can be seen in Figure 18b for the draft ratio as well, where the number of dissatisfied people is above 15% in some areas. In general, a location for an air heat pump that fulfills the Swedish criteria for velocities below 0.15 m/s is hard to find.

![Figure 18. Air/Air heat pump for building 2, a) operative temperature distribution at 0.1 m, b) draft ratio at 1.1m above the floor level.](image)

6.4.3 Air heating

6.4.3.1 Displacement devices

Overall a uniform horizontal operative temperature distribution appears in the simulations for the air heating system. Some warmer and colder regions are found close to the air inlets. See Figure 19a, for the operative temperatures at the 0.1 m level. The vertical operative temperature gradient for this case is around 1.2°C. Areas with high air velocities were found close to air inlets and door openings. At these places there were values that exceeded the 0.15m/s limit. For this building, the operative temperatures are in the limit for fulfilling the requirements for using an air heating system. The high air velocities could probably be avoided by placing the supply air devices outside the occupied zone, and using air transfer devices between the different rooms.
Results and Discussion

Figure 19. Air heating system for building 2, a) operative temperature distribution at 0.1 m, b) draft ratio at 1.1m above the floor level

Figure 20. Vertical Operative temperature gradient for building 2, the location of the line is illustrated as L1 in Figure 6.

6.4.3.2 Mixing ventilation devices
For the Building 3 case with ceiling ducts, the simulation is not comparable with building 2, since this building consists of two floor levels with buoyant flow and a forced flow through the staircase opening. For this case, two different airflows were studied. First with the actual airflow inside the building, designed with supply air in each of the bedrooms and living rooms with flows designed from standard values and the second was with rearranged airflows. For the actual airflow case around 2/3 of the heat is supplied at the top floor and 1/3 of the heat at the bottom floor. The results for temperatures and PPD for this case are presented in Figure 21.

This case gave a temperature gradient between the different floor levels of 3°C at 1m above the floor level at each floor and a local vertical temperature gradient inside the occupied zone between 0.8-1.7°C. The PPD values for the plane in the middle of the building shows that local thermal discomfort emanates from the cold draft on the ground floor and too high temperatures on the first floor.
In order to achieve a better indoor climate, the airflow distribution was changed so that the main part of the heat was supplied at the bottom floor. This gives an overall better indoor climate on the first floor and overall small temperature gradients between the different floors, see Figure 22a. But due to the high supply airflows in the living room and bedroom on the bottom floor, too high temperatures appear in these rooms, which results in poorer PPD values, see Figure 22b.

In the three rooms with closed doors the air temperature was always too low.

![Figure 21. Thermal climate for building 3 at design outdoor temperature. a) indoor temperature b) PPD (%).](image)

![Figure 22. Thermal climate for building 3 at design outdoor temperature with rearranged ventilation flows. a) indoor temperature b) PPD (%).](image)

On the whole the thermal indoor climate is better when a more evenly distributed heat is supplied to the building. Therefore, in general traditional systems with distribution of heat in every room of the building (underfloor heating and radiator system) achieve a good thermal climate for buildings with ventilation flows at minimum values at 0.35l/s m², according to Swedish regulations. But more locally distributed heating systems require a higher demand on the design in order to achieve a good thermal comfort. CFD of thermal
indoor climate is primarily a good tool for examining systems with heat not distributed in each room, like air heating, air heat pumps and a stove for example.

Decreased heat demand for a building entails a better thermal indoor climate. Air heating systems will therefore perform better in more insulated buildings. For two-story houses with an opening between the floors, the design of the heating system will be more demanding. The rooms with closed door and exhaust airflow do not achieve enough heat to produce a good indoor climate. These types of rooms should therefore be placed in the middle of the house. For air transfer between rooms using a gap beneath the door, air with the lowest temperature enters the room, which causes an even lower temperature.

6.5 Influence of snow for foundation heat losses

In a sub-arctic climate snow acts as an insulation layer against the soil for a large part of the year. In the standard method for calculation of heat losses the snow and ground freezing are not considered. In paper V the influence on heat losses from snow and ground freezing is studied for common foundation types used in Sweden. The study was performed in sub-arctic climate located in Luleå. In total four different foundation types were compared, see Figure 7. The work was done in order to investigate if snow has a significant effect on the heat losses from a building.

The year considered in the simulations was from 8th May 2014 to 8th May 2015 and the maximum snow depth was above 1 m, see Figure 25 for variations in snow depth. The simulation model was validated for the crawlspace case (Foundation A).

The validated results are presented in Figure 24. The maximum difference in air temperature between the simulated and measured crawlspace was 1.4°C with an average difference of 0.4°C Overall the model shows a good agreement with measurements. For the model, it was assumed that the snow close to the foundation at the soil had the same depth as the overall snow depth. The good agreement with measurements’ data indicates that the snow closest to the house cannot have melted during the winter caused by heat transfer from the building.
Figure 24. Simulated temperatures (black line) and measured temperatures (blue dashed line) in the crawl space.

The influence on the heat losses from snow and ground freezing was studied for all four different foundations with the model. The heat losses were calculated as heat through the subfloor of the building. A higher heat loss through the floor induces a larger temperature gradient inside the building except for underfloor heating systems. The yearly heat losses with and without the effect of snow and ground freezing are presented in Table 8. The overall heat losses were reduced by 7.1-10.0% if the snow layer and ground freezing were included in the simulations. The difference during the time of the year with snow coverage was 10.5-16.8%. It can be seen that snow has the largest impact on foundation type B and lowest for foundation type D at 7.1% over the year. The difference for foundation type B is due to heavily decreased heat losses from the thermal bridge around the lower part of the concrete beam that appears with a snow layer. For foundation Type D with extra insulation in order to reduce the thermal bridges, the heat losses have only a small influence from the snow layer.

On the whole foundation type D has the best performance with annual heat loss of 1033 kWh, and the foundation type A have the largest heat losses, 1852 kWh.
Table 8. Annual heat losses with and without the snow layer included.

<table>
<thead>
<tr>
<th>Case</th>
<th>Annual heat losses [total model] (kWh)</th>
<th>Annual heat losses [without snow and soil freezing] (kWh)</th>
<th>Difference (%)</th>
<th>Difference During winter (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>1852</td>
<td>2002</td>
<td>7.5</td>
<td>12.2</td>
</tr>
<tr>
<td>B</td>
<td>1133</td>
<td>1259</td>
<td>10.0</td>
<td>16.8</td>
</tr>
<tr>
<td>C</td>
<td>1432</td>
<td>1559</td>
<td>8.1</td>
<td>13.1</td>
</tr>
<tr>
<td>D</td>
<td>1033</td>
<td>1112</td>
<td>7.1</td>
<td>10.5</td>
</tr>
</tbody>
</table>

In Figure 25 the yearly variation with and without snow and ground freezing is presented for all the different foundations.

Figure 25a shows the difference for the temperature inside the crawlspace. Including snow and ground freezing increases the temperatures inside the crawlspace by 4°C. In the Figure the temperatures for a case without snow but with ground freezing are also presented. It can be seen that snow has the majority of the impact for the heat losses, while ground freezing only has a minor impact.

In Figure 25 b-e, the snow has the largest impact on the heat losses for the foundations A-C during the end of December to the end of February, while for type D the snow affects the heat flux for a longer period due to smaller sideward losses.

On the whole the study of different foundation types shows that snow must be included in a sub-arctic climate to predict the heat losses more accurately from a building foundation. The methods used in the common simulation software packages do not include snow as a parameter for ground heat losses.
Figure 25. Influence of the snow layer and soil freezing: a) temperature difference for the crawl space (foundation type A); b-e) heat flux with and without a snowlayer for the different foundations.
Results and Discussion
7 Conclusions

CFD models can predict the thermal indoor climate in a building. The validations show a good correlation between the measured and simulated temperatures and velocities.

The CFD method is suitable for investigating or designing systems with a heat supply in only a few rooms of the building. In these cases, the heat transport is caused by airflows between the different rooms and floor levels. The validation shows that a CFD model can predict temperature gradients for a two-story building both in supply and exhausted airflow to rooms, with errors less than 0.2°C, inside the occupied zone.

A CFD model describing the indoor climate can be simplified with user-defined wall functions. The UDFs can reduce the number of cells and some parts of geometry can be simplified in order to shorten the simulation times without significant accuracy loss.

A grid edge size of around 0.1 m in a building is sufficient to get accurate results according to discretization errors.

Steady state simulations can be used to determine the thermal indoor climate for a winter case in a sub-arctic climate, since the solar radiation only has a minor effect. The thermal balance requires that the thermal inertia is included in the boundary conditions.

For the investigated heating systems, the underfloor heating system fulfills Swedish requirements. Air heating through the ventilation ducts and air/air heat pump systems has problems with fulfilling the requirement of air velocities below 0.15 m/s in the occupied zone.

The PPD and PMV values in the EN ISO 7730 standard are hard to reach with air heating through the ventilation system, as in building 3. The high values are caused by an uneven distribution of the heat supply between the different floor levels. Rearrangement of the air supply improves the thermal comfort. However, some overheating appears in some rooms. This investigation shows the benefits of using CFD as a design tool for a heating system.

Snow and ground freezing, common in sub-arctic climates, influence the heat loss of a building. Snow has the largest impact on the heat losses, while soil freezing only has a minor effect. When simulating buildings in a sub-arctic climate, snow should be included in order to correctly describe the heat losses.

The maximum reduction in heat loss when including snow and ground freezing was 12-25% for the investigated foundation types. Annual heat losses were reduced by 7-10%. For the studied foundation types, the results show that insulation of the vertical sides of the construction has a major impact on the heat losses.
Conclusions
8 Future work

It has been validated that CFD can predict the temperature variation for a locally distributed heating system. However, this method requires more work in order to improve the indoor climate and the distribution of airflows and how a system should be designed when there are closed doors. The location of the supply air devices, mass flows and supply temperatures, and the usage of air transfer devices should be studied in order to find a good method to design the air heating systems with a good thermal comfort. The air heating systems may not be designed with supply air in only bedrooms and living rooms for a good thermal comfort. The distribution must also be designed according to heat demand.

More locally distributed heating systems will be installed in the future, due to their lower price and the fact that more energy efficient buildings will be constructed. In order for the construction industry to use CFD software in the design process, more studies with simplified models must be performed, where user friendliness and simulation time are studied.

For the soil model, future work needs to include moisture transport in order to simulate the relative humidity in crawlspaces, which avoids mold problems. This kind of model could be used as a design tool for ground constructions.

The thermal model to study the effects of snow can be used to investigate building constructions. The roof may, for some constructions, be heavily influenced by the snow, especially for warm roof constructions. However, ventilated roofs will also be affected by snow. The snow layer will have a positive effect on the thermal losses of the building.
Future work
9 References


[40] L. Peeters, I. Beausoleil-Morrison, A. Novoselac, Internal convective heat transfer modeling: Critical review and discussion of experimentally derived correlations,
References


References

[80] Z.J. Zhai, W. Zhang, Z. Zhang, Q.Y. Chen, Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: part 1 -


References


References


Paper I
COMPUTATIONAL FLUID DYNAMICS SIMULATION OF INDOOR CLIMATE IN LOW ENERGY BUILDINGS

Computational Set Up

by

Daniel RISBERG*, Lars WESTERLUND, and J. Gunnar I. HELLSTROM

Lulea University of Technology, Lulea, Sweden

Original scientific paper
https://doi.org/10.2298/TSCI150604167R

In this paper CFD was used for simulation of the indoor climate in a part of a low energy building. The focus of the work was on investigating the computational set up, such as grid size and boundary conditions in order to solve the indoor climate problems in an accurate way. Future work is to model a complete building, with reasonable calculation time and accuracy. A limited number of grid elements and knowledge of boundary settings are therefore essential. An accurate grid edge size of around 0.1 m was enough to predict the climate according to a grid independency study. Different turbulence models were compared with only small differences in the indoor air velocities and temperatures. The models show that radiation between building surfaces has a large impact on the temperature field inside the building, with the largest differences at the floor level. Simplifying the simulations by modelling the radiator as a surface in the outer wall of the room is appropriate for the calculations. The overall indoor climate is finally compared between three different cases for the outdoor air temperature. The results show a good indoor climate for a low energy building all around the year.

Key words: CFD simulation, computational set up, indoor climate, low energy buildings, turbulence models, grid size, radiation, convection, radiator, near wall treatment

Introduction

Investigation of the indoor climate in buildings can be performed with CFD technology. The air velocity and temperature can be predicted in all parts of the building with these simulations. Many studies have been made with the aim of predicting the indoor climate and heat losses in buildings using CFD software. Studies concerning computational set up for an entire low energy building are however missing in the literature. The first use of CFD to predict indoor airflow was made in 1974 [1]. In the last decades computer performance has increased rapidly and even extensive amounts of data are possible to handle with personal computers today. Together with improved software applications an increased use of CFD has occurred. The heat consumption for single-family houses (low energy buildings) decreases each year with more compact and better building envelope constructions. The concept low energy house refer to a building where the heat demand and peak load is significantly decreased compared to a building constructed according to Swedish National Board of Housing, Building and Planning’s recommendations (BBP), different examples are Passive houses, Zero energy houses, and

*Corresponding author, e-mail: daniel.risberg@ltu.se
Mini energy houses [2]. A study evaluation of the 9 passive house in Sweden has shown that actual energy consumption on is 21-35 kWh/m² floor area [3] compared to around 80 kWh/m² for a conventional building.

Traditional methods of heating and ventilation may be overambitious in order to maintain a good indoor climate when the heat consumption is decreased. New efficient methods can maybe result in a similar indoor climate with less investment cost. Most of the published work with CFD simulations is focused on individual rooms with quite a high rate of supply air or using natural ventilation [4, 5]. A study regarding office ventilation schemes have simulated the indoor climate with an air change rate at 4 times [6]. In this study the flow rate is more realistic for single family houses at (0.35 l/s per m²) which are common standard in BBP. Simplifications such as to include the radiator in the outer wall and convective heat transfer coefficients through specified equations has been implemented, all in order to decrease the computational time. This has not been published earlier. Also simulations concerning a low energy building in subarctic climate is new.

Summarizing of different studies regarding turbulence models for both indoor and ventilation simulations have earlier been performed [7]. The conclusion of the paper is that many turbulence models have been used for indoor-air simulations and no universal model for turbulence can be chosen. The choice of turbulence model is mostly dependent on accuracy and time needed. Studies like [8] compare different k-ε models where random number generation (RNG) show slightly better results as compared to the standard k-ε.

The heat losses through a building envelope are dependent on the conduction through the element, convection on both sides of the construction and radiation to/from the surfaces. The conduction part of a building element is easy to predict in a good way, but in order to predict an accurate model for indoor air-flow and temperatures the convective heat transfer is a difficult issue. The convective heat transfer coefficients for the isolated building surface are dominated by natural convection, and these values are hard to predict correctly both in CFD models and in experiments. There are many studies that have performed measurements of the heat transfer coefficient with large variations in the results. The study [9] concluded that the errors in his results in the best case are around 15%, [10] show that the mean values are around 1.4 W/m²K for a wall with no heating and around 1.6 W/m²K for a continuous heated wall. A review of empirical formulas for different isolated building surfaces has been published by [11].

The aim with this work was to establish computational set ups for simulations of the indoor climate in low energy buildings.

**Methodology**

In low energy buildings the convective forces are reduced since the heat transfer rates are so low when compared to ordinary buildings. This may affect the indoor climate in the building. Therefore, the volume of a one-family house is huge simplifications and a quite coarse mesh has to be introduced in order to make the simulation time more reasonable. A two room model was used in order to reduce the simulation time for the different tests to find the most suitable simulation set up. The elaborated set up was then finally used for an up scaled model of a whole house, this simulation was validated through measurements.

**Simulation object**

The simulated volume has two equal rooms with dimensions \( W \times L \times H = 2.0 \times 4.0 \times 2.5 \) m connected through an opening (door), fig. 1. One room is ventilated with supply air, and the air-flow is then transferred to the other room where it leaves as exhaust air. The ventilation
ducts are placed close to the ceiling in the middle of each room. The heat supply through the radiators is simulated as a heat flux through the envelope face \( W \times H: 1.0 \times 0.6 \text{ m} \) placed as one part of the short side of each room, a window \( W \times H: 1.0 \times 1.0 \text{ m} \) is placed above the radiator and simulated in the same way as the radiator. The supply air-flow rate is equal to 0.35 l/s for every square metre of floor area, which is around 0.5 air change per hour (ACH). The value is according to BBP. In a room the occupied zone (volume people normally exploit) is defined as the occupied zone is enclosed by two horizontal levels, one 0.1 m above floor level and the other 2.0 m above floor level, and a vertical level 0.6 m from the exterior wall or other external limit, or 1.0 m by windows and doors [12]. The indoor climate according to BBP for operative temperature is at least 18 °C inside the occupied zone with a maximum difference of 5 °C inside this area and a highest temperature at 26 °C. Also the air velocities have a limit of 0.15 m/s for the winter season and for the relative humidity (RH) the critical value is 75% RH [12] in the occupied zone. A further demand is that the floor temperature in the building should not be lower than 16 °C. A vertical line from the floor to the ceiling was created in the room with exhaust air, fig. 1. The position was chosen to be in the occupied zone and where the influence from window and radiator was expected to be strongest. Results from simulations are presented along this line.

The overall heat transfer coefficient, \( U \), for the different building elements in a low energy building was manually calculated and representing of a typical low energy building in Sweden. The \( U \)-values are for outer walls 0.1 W/m²K, ceilings 0.085 W/m²K, ground 0.15 W/m²K, and windows 0.85 W/m²K. The simulations were made for three different cases of the outdoor air temperature for the Swedish town Lulea, according to Swedish Meteorological and Hydrological Institute, winter (−30 °C), summer (±20 °C), and for an annual mean outdoor temperature (±2 °C). This was done in order to examine the indoor climate during a year. The supply air temperature was set at +18 °C, a design temperature for a mixing supply device in the ventilation system. The room temperature was designed for a temperature of +20 °C. The transmission losses through the different building elements were calculated by multiplying the overall heat transfer coefficient with the air temperature difference. Other heat losses caused by infiltration through the envelope due to a pressure difference between the indoor and environmental conditions, it was set at a constant value of 0.1 ACH. Furthermore, heating of the supply air from inlet temperature to room temperature is made by the radiator. The summation of all heat losses in each room was calculated and applied as a positive heat flux from the radiator to obtain the design indoor temperature.

**Numerical method**

The numerical simulations were performed with the commercially available software ANSYS CFX 14.5. The programme solves partial differential equations numerically with a control volume based finite element method. The governing equations that are solved for are the conservation of mass, momentum, and energy. The discretisation method used for the simulations was a second order upwind scheme. For the turbulence numeric, first order upwind has been used in order to ease convergence. A convergence target of 1e-7 for the scaled root mean square residuals was used for all the governing equations and turbulence model equations. For the summer case a value of 1e-4 was used. The small value was chosen since the air temperature

![Figure 1. Simulated volume fin used in this study](image-url)
changed very slowly. The buoyancy effect was modulated according to the Boussinesq method, which uses a correlation where the density difference is calculated according to eq. (1), with a constant reference density $\rho_{ref}$ and buoyancy reference temperature $T_{ref}$ [13]:

$$\rho - \rho_{ref} = -\rho_{ref} \beta (T - T_{ref})$$  \hspace{1cm} (1)

The density of the air was also modelled as an ideal gas in order to investigate the influence from the chosen method.

Turbulence models and near wall treatment

For indoor conditions the air-flow is generally turbulent also for cases with low velocities [14]. Turbulent flow is difficult to solve directly, hence the use of a CFD software that utilizes a turbulence model to predict the fluctuations in the air motion. A common model is the $k-\varepsilon$ model, which is a two equation model where one predicts the turbulent kinetic energy, $k$, and the other one predicts the eddy dissipation, $\varepsilon$. The standard $k-\varepsilon$ method is a widely used turbulence model for these kinds of applications, which predicts the air-flow quite well in the simulation domain and is a good compromise between simulation results and computing time [13]. The RNG $k-\varepsilon$ model is better adapted for flows with low Reynolds number because of an improved consideration of the effective viscosity at low Reynolds number [15]. The deviations between the two models are different constant values in the equations for described variables above and that in the turbulence equation standard the $k-\varepsilon$ model uses a constant $C_k$, whilst the RNG $k-\varepsilon$ model uses a function $C_{kRNG}$.

Two $k-\omega$ turbulence models, Standard and SST, were also introduced in the investigation of turbulence models. The $k-\omega$ models calculate the turbulent kinetic energy, $k$, together with the $\omega$ equation, which predicts the turbulent frequency. The SST model combines the $k-\varepsilon$ and $k-\omega$ models utilizing the good behaviour of each model. The SST uses the $\varepsilon$ formulation in the free stream and the $\omega$ formulation close to the wall. For the SST model a third wall scale equation is added.

The buoyancy was incorporated in the production term as a source term for the turbulent kinetic energy equation in all investigated turbulence models. Excluding buoyancy from the kinetic energy equation was also investigated in order to find the effects of its implementation. With the production term included a transient behavior appears, due to this a time dependent simulation had to be performed. This leads to long simulation times with only small differences in the results compared to neglecting the production term, mostly near the boundary regions. Therefore, in the simulation the standard setting was to exclude the buoyancy turbulence terms in order to get a stable solution.

The wall functions used to predict the near wall flow are the scalable wall functions. The improvement from the standard log law is that the singular separation points appear where the near flow velocity becomes zero, since the scalable wall function switches from a logarithmic approximation to a linear one for a $y^+$ value (dimensionless distance from the wall) of 11.06 [13]. In scalable definition the $u_*$ is replaced by $u'$, which describes a scalable velocity, in addition to scaling the $y^+$ value to a minimum value of 11.06. For the simulations with $\omega$-based formulation the wall function chosen is the automatic wall function [13], since the $\omega$-formulation requires a better resolution of the near wall region. In order to fully utilize the benefits of the SST-model a $y^+$ value close to 1 is recommended.

The convective heat transfer between wall and indoor air was treated by the software according to chosen wall functions. For the scalable wall treatment the convective heat flux was calculated:
\[ q_v = \frac{r_e u^*}{T} (T_w - T_r) \]  \hspace{1cm} (2)

**Boundary conditions**

The boundary conditions are difficult to set properly in air-flow simulations. In this study the inlet of the supply device for the ventilation was modulated with a constant velocity of 2.23 m/s (which represent an air-flow of 0.35 l/s for every square metre of floor area) perpendicular to the inlet face, fig. 1. The outlet was set as a pressure outlet with constant pressure equal to the atmospheric pressure. Infiltration to the volume was neglected in the simulation due to the conservation of the mass in the software. The heat losses from this phenomenon were however included in the calculations and evenly distributed on envelope surfaces.

The envelope face was expressed with a no-slip condition and the thermal boundaries were described as a fixed heat flux on the faces based on \( u \)-values presented in section *Simulation object*. The values was set according to tab. 1, for the radiator the heat flux is represented for a simplified radiator surface. The radiation model is very important for the prediction of both the temperatures and the velocities in the flow domain [16]. Simulations with and without the radiation model were performed in order to investigate its influence. The P-1 radiation model [13] was used between the internal building surfaces. Inner walls were simulated as adiabatic and for all walls an emissivity of 0.9 was chosen. Window surfaces emissivity was set at 0.5, which represents a value of a low emissivity window on the market. For the radiators face an investigation with and without radiation was performed and an emissivity of 0.1 was used when radiation was activated. The low value used is due to the fact that the area of the radiator face was modulated smaller than its real size and the high surface temperature determined by the software. The radiant effect used was about 30% of total heat transfer rate according to Persson [17]. Investigation of simplifying the radiator as a face in the envelope surface or modelled as a real radiator located in the room was made with and without radiation involved.

For the summer case the solar radiation through the window was modulated as a heat flux that is representative of the solar load and placed at a direction in 45° towards the floor from the south window. The solar heat flux was calculated:

\[ q_s = AG I_{sol} \sin(\phi) \]  \hspace{1cm} (3)

According to this correlation the window directed to the south (positive z-direction in fig. 1) has a radiation heat flux of 100 W/m² with \( G \) of 0.5 and the window area is reduced to 0.8 m² because of area hidden from the frame.

**Grid verification**

For indoor climate simulations, as for all simulations the goal is to get a grid independent solution [14]. A small change in results during a grid refinement is almost impossible to avoid. Therefore, a grid dependency study was performed for the model, using the grid convergence index (GCI) together with an extrapolated curve according to Richardson extrapolation,
which was used to define the discretisation error [18]. The investigated grid sizes were chosen in order to get an almost constant value for the refinement ratio, \( r \). Refinement of the mesh is done by minimizing the maximum size of the element in order to achieve a constant refinement rate. With use of an unstructured grid the ratio can be calculated according to eq. (4), following the approach by [19]. The parameter \( D \) describes the dimensions in the simulation, hence \( D = 3 \) in this case. The number of elements, \( N \), depends on the grid size and the simulation geometry:

\[
r = (N_{\text{fine}} / N_{\text{coars}})^{1/3}
\]

Investigated grid sizes with corresponding number of elements and a refinement ratio of \( r = 1.41 \) are: grid \( 1 = 883k \) elements, grid \( 2 = 314k \) elements, and grid \( 3 = 111k \) elements. Inflation layers (5 rows with a total depth of around 0.25 m) were used to resolve the convective air-flow close to the different surfaces. The zones close to the ventilation devices were meshed with a finer grid.

The most interesting variables for the indoor climate, velocity and temperature, respectively, were compared along the chosen line between floor and ceiling in the room with exhaust air, fig. 1. The difference \( \gamma \) in variable value \( \phi \) was calculated according to eq. (5), \( \phi_i \) is the solution for the finer grid and \( \phi_2 \) for the coarser grid. The same relationships were used for calculating \( \gamma_{21} \):

\[
\gamma_{21} = \phi_2 - \phi_1
\]

The order of GCI \( p \) was calculated:

\[
p = \frac{1}{\ln(r_{21})} \left[ \ln \left( \frac{\gamma_{22}}{\gamma_{21}} \right) + q(p) \right]
\]

The parameter \( q(p) \) in eq. (6) is equal to zero when the grid refinement ratio \( r_{21} \) equals \( r_{32} \). When the value of \( \gamma_{22}/\gamma_{21} \) becomes less than zero, oscillating convergence appears, but if the value of \( \gamma_{22} \) or \( \gamma_{21} \) is nearly zero a sign of a mesh independency has been reached. An additional grid refinement was made when both \( \gamma_{21} \) and \( \gamma_{22} \) were nearly zero, which gave a total number of elements of 2.4 million elements. A global order of \( p \) was calculated as average value of all local values of \( p \) along the investigated line. This value of \( (p) \) was used in all further calculations. The approximated GCI value was calculated from eq. (7). The expression gives the error of the variable value for the finest grid in actual point:

\[
GCI_{21}^{\text{line}} = \frac{1}{2} \frac{1.25e^{2}_{\phi}}{r_{21}^{2} - 1}
\]

The approximate relative error, \( e_{\phi}^{21} \), was calculated according to:

\[
e^{2}_{\phi} = \frac{\phi - \phi_2}{\phi_2}
\]

For the Richardson method an extrapolated solution \( \phi_{\text{ext}}^{21} \) was calculated for an infinitely fine grid according to:

\[
\phi_{\text{ext}}^{21} = \frac{r_{21}^{2} \phi_1 - \phi_2}{r_{21}^{2} - 1}
\]
Measurements and validation

The chosen methods from the result section was validated for a single family building where measurements was performed during two months during March and April 2014. The sampling time was carried out on minute basis. The validated values carried out at an outdoor temperature of +6 °C. Five temperature sensors was placed according to the fig. 2 at a height of around 2 meters above the floor level. The sensors were of type pt100 and according to manufacture measurement errors are in the ranges of ±0.4 °C. The heat supply through a water heating system was measured with a Kamstrup, Multical 402 device with an accuracy of 2%. The U-values of the building elements was the same as described for the two room model, at current outdoor temperature heat flux values according to tab. 2 was established.

Results

The results were compared along the vertical line illustrated in fig. 1, since it gives a good representation of the occupied zone. The velocities have a value slightly higher than the mean velocity and the temperatures show a good representation of the average temperatures inside the house, with respect to different horizontal planes with varying heights.

Grid refinement study

A grid independency study was performed for the case with mean outdoor temperature. Furthermore, the Boussinesq approximation, the standard k-ε turbulence model, scalable wall functions, a second order discretisation method and the radiation model included were used. Figure 3 show the velocity magnitude and fig. 4 the temperature for the different grid sizes chosen. The values close to the ceiling in fig. 3 depend on convective heat transfer. The velocity plot shows only small differences for the investigated grid sizes. The largest deviation arises for the simulation with the smallest amount of elements, fig. 3b. The local order of accuracy p for the velocity profile at different points is in the range 0.14-7.07 with an average of 2.55. The discretisation errors GCI from eq. (7), according to the velocity have a maximum value of about 20%. This high value is relative to a low velocity and corresponds to a maximum uncertainty in velocity of about ±0.015. The velocities are below the recommendation according to BBP in the occupied zone and should therefore not effectuate any discomfort in the indoor climate. The extrapolated curve fits to the case with 2400k grid cells.

For the temperature profile, fig. 4, the coarsest grid shows deviations compared to the other simulations. The remaining cases gave only small differences in temperature. The local order of accuracy is between 0.35 and 14.55 with an average of 7.11, see fig. 4b. The temperature values for the different grid sizes show oscillating convergence in 70% of the values. Since the difference in variable value γ for both γ21 and γ22 is nearly zero, an additional simulation was made with a refinement grid of 2400k cells. The variable temperature seems to show a grid independent solution for the case with around 314k elements and small discretisation errors. The extrapolated curve fits to the temperature curve for all cases except the coarsest case.

<table>
<thead>
<tr>
<th>Building element</th>
<th>Wall</th>
<th>Floor</th>
<th>Ceiling</th>
<th>Window/door</th>
</tr>
</thead>
<tbody>
<tr>
<td>Heat flux [W/m²] (+6 °C)</td>
<td>-1.92</td>
<td>-2.64</td>
<td>-1.5</td>
<td>-14.96</td>
</tr>
</tbody>
</table>
According to the grid refinement study a grid with face length around 0.1 m and with 5 inflation layers (the case of 314k elements) gave acceptable results. For larger volumes like one-family houses, the central processing unit time with this grid size will also be tolerable. Hence, all further work in this paper was performed using this grid size. Investigation of two different buoyancy models, the Boussinesq approximation and ideal gas law behaviour was performed. All other settings that were used during the grid size study were applied. Along the actual line the difference was between 0-0.007 m/s for the velocity and 0-0.1 °C for the temperature. This small deviation between the two models and a faster convergence for the Boussinesq approximation motivated the choice of this buoyancy model for all the further work.

**Turbulence models**

Four different turbulence models were compared in order to investigate the most relevant model for CFD simulations of indoor climate. The turbulence models that were compared are widely used for indoor air-flow calculations. The same settings as used earlier were applied. Investigated turbulence models were two $k$-$\varepsilon$ models, Standard and RNG, and two $k$-$\omega$ models, Standard and SST. Velocities and temperatures are presented for the different turbulence models in fig. 5. The results show that the velocities deviate among the different models, while for the temperature the standard $k$-$\omega$ model creates the results that deviates the most.

The velocities in fig. 5 are lower than the recommendation by BBP, inside the occupied zone. The deviation among the models will therefore not affect the indoor climate, from a human perspective. The standard $k$-$\omega$ model shows the largest deviation and was therefore excluded.
All other models are possible to use for CFD simulation of the indoor climate in buildings, since they capture the main flow features for convective heat transfer from a radiator according to experiments [20, 21]. Even the hot negative buoyant jet that appears on the opposite wall from the radiator is detected, see fig. 9. The results from the standard \( k-e \) and \( k-\omega \) SST show the smallest difference relative to each other. The average \( y^+ \) for the simulations is in the range of 9-11 with are far from the recommended values for the standard \( k-\omega \) and the SST model. The computing time and convergence rate are fast for the standard \( k-e \) model and it is well used in all kinds of applications, hence this model was chosen for all further work.

![Figure 5. Velocity and temperature profile for the different turbulence models; (a) velocity, (b) temperature](image)

**Radiation effects**

The effects of radiation in the simulation model are very important even though the temperature differences between the surfaces are small. Figure 6 shows the temperature profile along the investigated line for two cases with mean outdoor temperature boundaries, with and without radiation involved (radiator surface excluded). For the case without radiation the temperature drops to an unreasonably low value at the floor level. The velocities are very small and therefore more dense cold air is stacked up in this area. The indoor air temperature and velocity are almost the same at the other levels.

![Figure 6. Surface and air temperature with and without radiation model included](image)

**Radiator modelling**

Due to the irregular shape the radiator can not be meshed in its real shape since it would result in too many elements in the total volume of a house. A simplified radiator model, a face on the outer wall, was compared with a rectangular box \((W \times H \times D; 1.0 \times 0.6 \times 0.05 \text{ m})\) placed 0.05 m from the outer wall inside the room. Only convective heat transfer was assumed for these cases, and comparison where radiation was also included for both cases (30% of total heat flux) was performed. The air velocity and temperature along the line for the different cases are presented in fig. 7. A difference in velocity for the models appears, but the overall velocity is still small when compared with BBP. For the rectangular box with radiation the lowest velocities
appear due to the lower rate of convective heat flux from the radiator. A larger area compared to the face approach and radiation contribution is included. For the temperature the difference is small between the models but a smaller gradient appears for the rectangular box with radiation included. It can be concluded that modelling the radiator as a surface on the outer wall is sufficient when simulating the indoor climate in buildings.

Figure 7. Air velocity and temperature along studied line for the different radiator models

**Different environmental conditions**

The indoor climate was simulated for three typical cases of the outdoor conditions representative of the north part of Sweden, with settings according to the previous study. A vertical plane in the middle of the room with supply air, going from the window to the opposite wall and from the floor to the ceiling, is presented in figs. 8-10. Planes located in the other room and at other positions indicate equivalent results.

The winter case with an outdoor temperature of -30 °C, fig. 8, shows the highest air velocities above the radiator and along the ceiling towards the air supply unit. The highest air temperatures are also found in this area all due to buoyancy. The vertical temperature difference in the occupied zone of the plane is only 1.86 °C. This value is a good representation of the temperature gradient for the total volume of the two rooms. The air velocities in the plane show good mixing of the air and only low velocities. The heat supplied by the radiators was 245 W and with an area of 0.6 m² for each radiator.

Figure 8. Conditions at a plane in the inlet room during the winter case; (a) velocity vector field and (b) temperature distribution

The indoor climate for the mean outdoor temperature case is presented in fig. 9. The radiators supply 95 W each for this case. The air movements show a similar pattern as for the winter case, but the magnitude of the velocity and air temperature difference is somewhat lower
than for the winter case because of the reduced heat exchange between the surfaces and the indoor air. The comparable temperature difference inside the occupied zone is 0.77 °C in the plane.

For the summer case the heating system is turned off, and the solar radiation is modelled a one part that describes the direct solar radiation. The direct solar radiation is calculated from eq. (3) and applied at the floor with a direction of 45° from the south window with an intensity of 100 W/m². The summer case is presented in fig. 10. The velocities inside the occupied zone are low and the air shows a good mixing. The temperature gradients in the plane show very small variations, 0.1 °C inside occupied zone of the plane, and the air temperature increases inside the simulation volume to a value of 24 °C in the occupied zone. The temperature is lower than 26 °C, which is the recommendation by the Swedish Government.

![Figure 9. Conditions at a plane in the inlet room during the mean outdoor temperature case; (a) velocity vector field and (b) temperature distribution](image)

![Figure 10. Conditions at a plane in the inlet room during the summer case; (a) velocity vector field and (b) temperature distribution](image)

Average air temperatures in the room were calculated in horizontal planes every 0.5 m from the floor to the ceiling. In order to avoid the influence from the radiator instead of the level 0.5 m two levels 0.1 and 0.75 m above the floor were chosen. The mean surface temperature for the floor and the ceiling was also included in fig. 11. The temperature gradient inside the room is steeper for the winter case, which is what one would expect due to the increased heat transfer through the external surfaces. The deviation arises at the floor surface temperature and in the area up to 1.5 m above the floor.

![Figure 11. Average temperature gradient for two cases with different outdoor temperature during the heating season](image)
For this low energy building the $U$-values are small and the temperature difference between floor and ceiling is low. The temperature gradient in the occupied zone is 1 °C/m for the winter case and 0.4 °C/m for the mean outdoor temperature case.

**Validation**

Measurements in a low energy building with one family resident have been performed. The building was located in the northern part of Sweden. To minimize the influence of the inhabitants the validation was done at early morning at 5 a.m. The air temperature in one point in each of the five rooms was measured. The outdoor temperature was +6 °C. Figure 12 shows measured and simulated values according to chosen CFD model of the air temperature in each room. Room 1 is the open planned living room and kitchen, rooms 2, 3, and 5 are bedrooms, and room 4 is the toilet. The difference is at maximum 0.75 °C, the simulated values are not including internal heat from the people sleeping in rooms 2, 3, and 5.

**Discussion**

In CFD simulations very specific problems are often studied, with large gradients in temperature and velocities and small grid cells are therefore normally used. When an entire building is to be simulated, an acceptable result can be established also for a coarse grid solution, since the gradients are small. A course grid is also necessary because of the large volumes investigated.

The scalable wall functions calculation of the surface temperature for the radiator seems to be high, but the air temperature in the first node point is feasible. A high surface temperature also causes an emissivity value that is fairly low in order to not overestimate the heat transfer mechanism.

A radiation model is very important to include in the simulation. The effect shows mainly on the floor temperature. Without radiation the floor temperature decreases to unreasonably low values due to buoyancy and low convective heat transfer rate. To increase the indoor climate the temperature on the floor and up to a level of 1.5 m above the floor should be raised.

With higher temperatures in this area the indoor climate can probably be preserved even if the air temperature inside the room decreases, i.e., the heat losses can be decreased even more.

**Conclusions**

The CFD simulation can be used to predict the indoor climate in buildings. For simulations of the indoor climate it is important to capture the main flow velocities and temperatures in the building.

A grid independency study shows that an acceptable number of elements is around 300k elements for the chosen simulation volume. This value represents a grid edge size of around 0.1 m to get an accurate result according to the discretisation errors.

The buoyancy effect must be included in the simulations of the indoor climate, and the Boussinesq approximation fulfills that demand. A comparison of different turbulence models shows quite small deviations, although the standard $k$-$\omega$ model shows the largest deviation. The standard $k$-$\varepsilon$ model should be used since it is numerical stable and gives good results with reasonable central processing unit time.
The velocities are much lower in the occupied zone as compared to the maximum acceptable value according to BBP, therefore the deviation between the models will not affect the indoor climate from a human perspective.

Radiation between building surfaces have a substantial impact on the temperature level for indoor climate simulations. Simulations with radiation excluded show too low temperatures on the floor. For a simulation with the mean outdoor air temperature a floor temperature around 12 °C was obtained, while with radiation included a floor temperature of around 19 °C was achieved. Radiation from the heat source has a small effect on the temperature gradient inside the occupied zone. However, the gradient in the room becomes straighter with radiation included in comparison with only involving convective heat transfer for the radiator. Simplifying the simulations by modelling the radiator as a surface in the envelope of the room is appropriate for simulations of the overall indoor climate.

Simulations of the climate inside the room with environmental winter conditions of -30 °C show low air velocities and temperature differences of less than 2 °C. When the outdoor temperature increases to +12 °C the indoor air temperature difference is only 0.8 °C. The temperature gradient inside the occupied zone decreases with increasing outdoor air temperature, which would be expected. The small difference originates from small heat fluxes through the envelope (low energy building). For the summer case (outdoor air temperature +20 °C) with direct solar radiation no temperature gradient appears inside the occupied zone due to the direct solar radiation and is detected by the floor. The average indoor air temperature was +20 °C except for the summer case where it rose to +24 °C. Future work in which the results of this work are incorporated, is to be implemented for a total building and validated in detail. To connect to the residents also comfort parameters will be studied in order to show the powerful tool that CFD constitutes.

Acknowledgment

This work has been carried out thanks to funding from the County Administrative Board of Norrbotten.

Nomenclature

\[ A \] – surface area, \([\text{m}^2]\)

\[ ACH \] – air change per hour, \([\text{h}^{-1}]\)

\[ cp \] – specific heat capacity, \([\text{kg}^{-1}\text{°C}^{-1}]\)

\[ D \] – dimensions in the simulation volume, \([\text{m}]\)

\[ e_{str} \] – relative error

\[ G \] – transmission of solar radiation through the window

\[ GCI \] – grid convergence index

\[ I_{rad} \] – solar radiation intensity, \([\text{Wm}^{-2}]\)

\[ k \] – kinetic turbulent energy, \([\text{m}^2\text{s}^{-2}]\)

\[ p \] – order of GCI

\[ q_r \] – radiation flux through the window, \([\text{Wm}^{-2}]\)

\[ q_{eq} \] – heat flux, \([\text{Wm}^{-2}]\)

\[ r \] – grid refinement ratio

\[ T_1 \] – near wall fluid temperature, \([\text{°C}]\)

\[ T_{ref} \] – reference temperature, \([\text{°C}]\)

\[ T_w \] – wall temperature, \([\text{°C}]\)

\[ U \] – overall heat transfer coefficient, \([\text{Wm}^{-2}\text{°C}^{-1}]\)

Greek symbols

\[ \beta \] – coefficient of expansion, \([\text{°C}^{-1}]\)

\[ \gamma \] – difference between variable values

\[ \varepsilon \] – turbulent eddy dissipation, \([\text{m}^2\text{s}^{-3}]\)

\[ \rho \] – density, \([\text{kgm}^{-3}]\)

\[ \rho_d \] – reference density, \([\text{kgm}^{-3}]\)

\[ \phi \] – variable value

\[ \phi_{eq} \] – extrapolated solution

\[ \omega \] – turbulent frequency, \([\text{s}^{-1}]\)

References

[16] Sørensen, D. N., Nielsen, P. V., Quality Control of Computational Fluid Dynamics in Indoor Environments, Indoor Air, 13 (2003), 1, pp. 2-17
Paper II
CFD modelling of radiators in buildings with user-defined wall functions

Daniel Risberg*, Mikael Risberg, Lars Westerlund
Energy Engineering, Division of Energy Science, Luleå University of Technology, SE-971 87 Luleå, Sweden

HIGHLIGHTS
• This paper studies how to deal with natural convective heat transfer for a radiator in order to simplify the simulations.
• By adding user-defined wall functions the number of cells can be reduced considerably compared with the $k$-$\omega$ SST turbulence model.
• Compared to manufacturer data the error of the model is less than 0.2% for the investigated radiator height and temperature.

ARTICLE INFO
Article history:
Received 6 July 2015
Accepted 23 October 2015
Available online 3 November 2015

Keywords:
CFD modelling
Radiator
User-defined wall functions
Indoor climate

ABSTRACT
The most widely used turbulence model for indoor CFD simulations, the $k$-$\varepsilon$ model, has exhibited problems with treating natural convective heat transfer, while other turbulence models have shown to be too computationally demanding. This paper studies how to deal with natural convective heat transfer for a radiator in order to simplify the simulations, and reduce the numbers of cells and simulation time. By adding user-defined wall functions the number of cells can be reduced considerably compared with the $k$-$\omega$ SST turbulence model. The user-defined wall function proposed can also be used with a correction factor for different radiator types without the need to resolve the radiator surface in detail. Compared to manufacturer data the error is less than 0.2% for the investigated radiator height and temperature.

1. Introduction
The indoor climate in buildings is often modelled using building simulation software that predicts the indoor climate as well as the mixed airflow with a uniform velocity in each room. This method does not meet the requirements for detailed indoor climate analyses where it is necessary to predict the air velocity and temperature at different locations inside the room [1].

With CFD simulation technique the indoor climate can be predicted in the whole simulation volume. Therefore the velocity and temperature distribution in a building can be studied in detail [2]. Several studies of simulation of the indoor climate have been carried out but usually focusing on single rooms or specific areas inside a building [3]. These limitations are mainly caused by the amount of cells in the computational mesh being too huge for simulation of an entire building. One way to be able to simulate the indoor climate in a complete building is to simplify the model to reduce the number of cells in the simulation volume.

For indoor climate simulations the most common turbulence model used is the $k$-$\varepsilon$ model [4] with scalable wall treatment. Previous studies using this model have shown good agreement for predicting the airflow within the room [5] and prediction of forced convective flow. However, the model is not accurate for predictions of natural and mixed convective flows [6]. In buildings this phenomenon is the main driving force for the indoor airflow in most situations and is therefore important to predict correctly. The scalable wall treatment has been shown to underestimate the convective heat transfer coefficients [7]. An underestimation of the convective heat transfer coefficients will result in a large temperature difference between surfaces and the indoor air, which will result in an over-prediction of the radiation heat transfer from the surfaces. This phenomenon has a large effect on the temperature of the warmest and coldest surfaces inside the building, for example radiators and windows.

Other turbulence models have been used to simulate indoor climate with good agreement and in order to predict the mean velocity and temperature gradient where large eddy turbulence models show the most accurate results [8]. The main drawbacks of large eddy simulation are the long computational time and large memory requirements, and it is therefore not used for simulations of large volumes. The main reason for this is the treatment of the near wall region where a very fine computational mesh is required that goes $y^+ \approx 1$; furthermore the simulation needs to be time resolved. Also the $k$-$\omega$
SST model has shown good agreement with measurement for simulation of indoor airflow with heat transfer [9]. The k-ε SST model also needs γ ≥ values around 1 and will require a higher number of cells compared to the k-ε model, which allows higher γ ≥ values. Therefore the k-ε model can use a coarser mesh and the computational time will decrease significantly compared to other models. The aim of this work was to investigate if the k-ε model could be used as turbulence model in order to correctly describe the indoor climate for a room with radiators as heat distribution system. Another objective was to simplify the geometry of a radiator in the model to be able to decrease the number of cells in the simulation volume.

2. Theory

2.1. Radiator type

The most common type of radiator is the panel radiator consisting of one or several panels to distribute the heat to the room (see Fig. 1). Different kinds of radiators have a different amount of convective heat transfer, which depends on the numbers of panels and the area of the radiator. The amount of radiation and convection heat transfer for a typical panel radiator is presented in Table 1.

2.1.1. Radiator standard testing

A radiator in Europe is tested according to European Standard EN 442-2 [11]. The test is performed with a 1 metre wide radiator for each height. The supply water temperature to the radiator is set to be able to decrease the number of cells in the simulation volume.

In order to decide the heat transfer rate (\( \Phi \)) for a radiator with other widths and temperatures, a formula called the one exponent formula is used [12].

\[
\Phi = \Phi_0 h_b \left( \frac{\Delta T}{\Delta T_0} \right)^n
\]

where (\( \Phi_0 \)) stands for the determined heat transfer rate according to EN 442-2, and h is the width of the radiator in unit metre. The logarithmic temperature (\( \Delta T \)) is the temperature difference between the air in the reference point and the radiator mean surface temperature; it is calculated according to Equation (2). The exponent n in Equation (1) is the radiator exponent that usually is in the range of 1.2–1.4 for all types of radiators. The most common value is around 1.3 [13]. \( \Delta T_0 \) is the logarithmic temperature for the normal test case (water temperature 75/65 °C) with an air room temperature of 20 °C.

\[
\Delta T = \ln\left(\frac{T_{\text{supply}} - T_{\text{room}}}{T_{\text{return}} - T_{\text{room}}}\right)
\]

where \( T_{\text{supply}} \) is the water supply temperature, \( T_{\text{room}} \) is the water return temperature from the radiator, and \( T_{\text{wall}} \) is the air temperature at the reference point.

2.2. Convective heat transfer calculations

The main problem for convective heat transfer is to determine the boundary conditions at a surface exposed to a flowing fluid. The local Rayleigh number (\( R_a \)) is calculated as

\[
R_a = \frac{g \beta (T_{\text{wall}} - T_{\text{ref}}) x}{\nu \alpha}
\]

where g is the acceleration of gravity, \( \beta \) the thermal expansion coefficient, \( T_{\text{wall}} \) is the wall (surface) temperature, \( x \) is the vertical position on the radiator surface, \( \nu \) is the kinematic viscosity, and \( \alpha \) is the thermal diffusivity. The Nusselt number (\( Nu \)) is established by [14] according to Equation (4). The Prandtl number (\( Pr \)) describes the relationship between momentum diffusivity and thermal diffusivity.

\[
Nu = \frac{3}{4} \frac{Pr}{\frac{Pr^\frac{3}{2}}{\frac{2}{4}} \frac{Pr + 0.4}{4.953} Pr}
\]

The convective heat transfer coefficient (\( h_c \)) is calculated as

\[
h_c = \frac{Nu \lambda}{x}
\]

The fluid thermal conductivity (\( \lambda \)) and vertical position on the radiator surface are included when the heat transfer coefficient is determined. Air data for \( v, \nu \) and \( \alpha \) at different water supply temperatures are presented in Table 2.

Newton’s law of cooling [15] expresses the heat transfer rate due to convective heat transfer as

\[
Q = h_c A (T_{\text{wall}} - T_{\text{ref}})
\]

where (A) is the area of the surface.

---

**Table 1**

<table>
<thead>
<tr>
<th>Type</th>
<th>Radiation heat transfer [%]</th>
<th>Convection heat transfer [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Single panel</td>
<td>50</td>
<td>50</td>
</tr>
<tr>
<td>Double panels</td>
<td>30</td>
<td>70</td>
</tr>
<tr>
<td>Triple panels</td>
<td>25</td>
<td>75</td>
</tr>
</tbody>
</table>

---

**Table 2**

<table>
<thead>
<tr>
<th>Water supply temperature (°C)</th>
<th>Mean difference temperature (°C)</th>
<th>( v ) (m/s)</th>
<th>( \nu ) (m²/s)</th>
<th>( \lambda ) (W/mK)</th>
</tr>
</thead>
<tbody>
<tr>
<td>75.0</td>
<td>47.5</td>
<td>17.97·10⁻⁵</td>
<td>27.83·10⁻⁵</td>
<td>25.56·10⁻⁵</td>
</tr>
<tr>
<td>55.0</td>
<td>37.5</td>
<td>16.97·10⁻⁵</td>
<td>27.09·10⁻⁵</td>
<td>24.08·10⁻⁵</td>
</tr>
<tr>
<td>45.0</td>
<td>32.5</td>
<td>16.46·10⁻⁵</td>
<td>26.72·10⁻⁵</td>
<td>23.34·10⁻⁵</td>
</tr>
</tbody>
</table>

---

Fig. 1. Radiator with two panels [10].

---

D. Risberg et al./Applied Thermal Engineering 94 (2016) 266–273
2.3. Numerical details

The $k$-$\varepsilon$ model is a two-equation model where one predicts the turbulent kinetic energy ($k$) and the other predicts the eddy dissipation ($\varepsilon$).

The wall functions used to predict the near wall flow for the $k$-$\varepsilon$ model are the scalable wall functions. The improvement from the calculations treatment uses the same expression for convective heat flux according to Equation (7). The different wall treatments induce differences in the $u^*$ and $T^*$ in different ways.

The $k$-$\varepsilon$ model with scalable wall functions calculates $T^*$ (the non-dimensional temperature) as

$$T^* = 0.1 \ln(y^+) + D$$

(8)

The variable $D$ is established through

$$D = (3.85 \Pr^{0.5} - 1.3) + 0.1 \ln(Pr)$$

(9)

The scalable velocity, $u^*$, for the model is calculated as

$$u^* = \frac{C_1}{\sqrt{\frac{u^*}{\nu}}} = (\frac{u^*}{k})^\kappa.$$  

(10)

$C_1$ is a turbulence model constant with a value of 0.09 and $\kappa$ is the turbulent kinetic energy.

The $k$-$\omega$ SST-model with automatic wall functions determines the variable $T^*$ as

$$T^* = \frac{\nu}{\rho} \frac{1}{y} \left( \frac{1}{y^2} + \frac{2.12 \ln(y^+)}{\beta} \right) \frac{\omega}{\nu}$$

(11)

where the variables $\beta$ and $\Gamma$ are determined as

$$\beta = (3.85 \Pr^{0.5} - 1.3) \Gamma + 2.12 \ln(Pr)$$

(12)

$$\Gamma = \frac{0.01 \Pr^{0.5} \nu^5}{1 + \Pr^{0.5} \nu^5}$$

(13)

The scalable velocity, $u^*$, for the $k$-$\omega$ SST-model is calculated as

$$u^* = \frac{C_1}{\sqrt{\frac{u^*}{\nu}}} = (\frac{u^*}{k})^\kappa.$$  

(14)

The relative tangential velocity to the boundary is denoted as $\Delta U = \frac{\Delta y}{\Delta t}$. $\Delta y$ is the distance to the first node and $\Delta t$ is the proportional-constant with standard value of 0.31 [16].

In order to improve the modelling of the heat transfer for the $k$-$\varepsilon$ model, user-defined wall functions (UDF) were implemented. The normalised wall heat flux coefficient ($w$) is calculated according to Equation (15), which represents the quota of the term $u^*/T^*$ in Equation (7). The heat transfer coefficient ($h$) is calculated according to Equations (3)–(5).

$$w = C_1 \cdot \frac{u^*}{T^*}$$

(15)

The theoretical expression of the convective heat transfer (Equation (6)) includes the temperature difference between the surface and the bulk temperature in the fluid. The software uses instead of bulk temperature the near wall temperature ($T_f$) in Equation (7).

In order to consolidate these methods, a reference point temperature of the air ($T_{ref}$ in Equation (6)) was established 1 m ahead of the radiator and 1 m above the floor level. A scale coefficient ($C_1$) used in Equation (15) was calculated as

$$C_1 = \frac{T_{ref} - T_{wall}}{T_{wall} - T_{wall}}$$

(16)

where $T_{wall}$ is the wall (surface) temperature and $T_f$ is the near wall temperature calculated at the first node value.

Finally to compensate for the simplification of the radiator surface and number of panels, a factor was introduced as $C_1$ in Equation (16).

The value of the constant is determined in the Methods section. For a radiator with a single panel and a flat surface, the constant is equal to one.

3. Methods

The commercial CFD code Ansys CFX 15.5 was used for the simulations. The work was divided into three steps according to Fig. 2.

In the first step the natural convection heat transfer for a flat plate radiator in 2D simulations was investigated through the methods described in the Theory section. The work was done in order to determine the most efficient method of the selected ones.

In the second step a 3D simulation of a real shape panel radiator was validated with the Standard EN-442-2 test. The model consists both of the airflow around the radiator, in the test room and the water flow through the radiator.

The third step included simplifications of the radiator geometry in order to find a good way to model radiators in CFD simulations of an entire building.

1) 2) 3)

2D Radiator

Investigate convective heat transfer coefficients for different turbulence models.

3D real radiator simulation

Validate the chosen model, with simulation of the En-442-2 test room.

Simplification

Find a way to model the radiator surfaces in CFD simulation of indoor climate.

Fig. 2. Work steps.
3.1. 2D radiator

The geometry for the 2D radiator case is presented in Fig. 3. The radiator was simulated with a constant heat flux of 200 W/m² on the front and back surfaces; the other radiator surfaces were treated as adiabatic. All surfaces in the room were modelled with the discrete transfer radiation model [17] and an emissivity of 0.9. For the opening surface (towards the remaining part of the test room) the boundary condition was modelled with a temperature of 20°C and a black body radiation of 20°C. For this geometry and boundary conditions the different setups discussed in the Theory section were compared: the \( k - \varepsilon \) model with scalable wall functions, the \( k - \omega \) SST model and the \( k - \varepsilon \) model with UDF.

3.2. 3D model of a real radiator

The \( k - \varepsilon \) model with UDF for heat transfer was compared to the manufacturer's data according to the Standard EN 442-2. The geometry for the EN 442-2 test room is presented in Fig. 4. The temperatures are specified for all surrounding surfaces in the room to give an air temperature around 20°C, except the wall behind the radiator, which was simulated adiabatically. The emissivity was set at 0.90. The radiator was modelled with three different water inlet mass flows and supply temperatures according to Table 3. The convective heat transfer between the water and the metal wall of the radiator was modelled with a high heat transfer coefficient. This is because the thermal resistance between the water and the air in the room mainly exists on the outside, between metal and air. The metal temperature will therefore be close to the water temperature. The emissivity was set at 0.95 for the radiator surface towards the air [18], and the wall function transfer coefficient was set according to Equation (15) for the vertical surface of the radiator. A reference point was set at 1 m in front of the radiator and 1 m above the floor level in order to establish the ambient air temperature. This was used for calculations of the heat transfer coefficient between the metal and air.

3.3. Simplifications

For CFD simulations of an entire building or a large room, the geometry must be simplified, since if every detail is modelled the number of cells becomes unrealistically high, which leads to long simulation times and large memory requirements. Especially the area close to the radiators must be simplified to reduce the number of cells in the simulation volume. A flat radiator surface will give an error when the heat transfer is modelled. Due to the reduced area of the radiator surface, an increased surface temperature will arise if a constant heat flux is set. Thereby the radiation part of the total heat flux will increase compared to the real case. If instead a constant temperature is set on the radiator surface, the total amount of heat transfer will be too low.

A way to compensate for this error is to insert a user-defined wall function where the wall heat transfer coefficient for the radiator surface against the air is adjusted to compensate for the decreased area of the radiator. Fig. 5 shows the difference in heat transfer rate for a panel radiator with different numbers of panels. The size of all radiators were \((W \times H)\); 1000*300 mm, the water supply temperature was 55°C, and the return temperature was 45°C with an air room temperature of 20°C. The markers represent data from a manufacturer [19]. An assumption was made that radiation between panel surfaces can be neglected (same temperatures) and only outer surfaces (front and back surfaces) directly contribute radiation heat to the room. Fig. 5 shows a straight line in heat transfer rate for a single and multi-panel (MP) radiator. Only the convective heat transfer part varies with the number of panels due to different areas for heat transfer between the radiator surface and the air in the room. From the equation of the straight line in Fig. 5, the amount of radiation heat transfer is represented by the constant \(90.13 \text{ W} \), and the gradient multiplied with the number of panels represents the convective heat transfer. The equation presented in Fig. 5 corresponds to the values in Table 1.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>75</td>
<td>0.00682</td>
<td>18.5</td>
</tr>
<tr>
<td>55</td>
<td>0.00442</td>
<td>19.1</td>
</tr>
<tr>
<td>45</td>
<td>0.00251</td>
<td>19.0</td>
</tr>
</tbody>
</table>

Table 3  

Boundary conditions.

Heat transfer rate = 90.203·MP + 90.13

Fig. 3. Geometry of the 2D radiator.

Fig. 4. Geometry of the 3D real radiator setup.

Fig. 5. Heat transfer rate for a radiator as function of numbers of panels.
The manufacturer also presents the heat transfer rate for a radiator with a flat surface towards the air and with the same size and conditions presented above. With a constant radiation heat transfer rate (90.13 W) for all cases, the convective heat transfer rate can be calculated as the total heat transfer rate with the radiation heat part subtracted. Using the flat radiator as reference case, the heat transfer coefficient was modified with a constant that describes the type of radiator used. The constant for the different radiator types is presented in Table 4.

The values for constant C2 in Table 4 are valid for the described conditions. Changing the water supply temperature or the height of the radiator will affect C2. For other conditions just follow the described method to obtain comprehensively correct values for the constant C2.

Simulations with a constant heat flux as boundary condition on the radiator surfaces do not correspond to a real radiator, since the heat flux varies with the highest values at the lower part of the radiator. A constant surface temperature was therefore set on the radiator surface calculated from Equation (2) and an air room temperature of 20°C. The heat transfer coefficient was calculated by the programme and in the user-defined wall functions adjusted with a constant according to Table 4. The convective heat transfer rate was calculated and the radiation heat transfer rate was also established. Adding these two parts gave the total heat transfer rate from the radiator. Simulations of three different cases were performed: flat radiator, one-panel radiator (MP1), double-panel radiator (MP2) and also an MP1 radiator with a height of 590 mm.

4. Results and discussion

4.1. 2D radiator

The total number of cells for the models is presented in Table 5, and the average y+ value at the radiator surface for each of the turbulence model was considered. For the case using the k-ε model with scalable wall treatment, a grid size of 40 mm cells gave a y+ value around 16 at the radiator surface and for the k-ω SST model a y+ value around 1. The number of cells needed for the k-ω SST is around 6 times higher than for the k-ε models in the total volume, and therefore the computational time will increase significantly for that case.

During the work it was found that the grid needed to be refined near the lower part of the radiator (the first 20 mm in height) for the k-ε model with UDF; a y+ value close to 1 was needed in this area in order to predict the heat transfer correctly. This is due to the rapid change in heat transfer coefficient in that area.

The calculated convective heat transfer coefficient and surface temperature for the radiator front area are presented in Fig. 6. The k-ε model with scalable wall functions predict a considerably lower heat transfer coefficient compared to the SST model. For the case with user-defined wall functions implemented, the predicted heat transfer coefficient corresponds well to the SST model.

The amount of heat transfer from convection and radiation for the different models is shown in Fig. 7. For the k-ε model with scalable wall functions, radiation contributes 73% of the total heat transfer rate, which is too high according to literature data. The high
surface temperature affects a large radiation part. The SST and the user-defined model show a radiation part around 50%, which is close to the values for a one-panel radiator presented in Table 1.

From the 2D simulation it can be seen that the k-ε model with scalable wall treatment underpredicts the convective heat transfer coefficients for natural convection, which results in a higher surface temperature and a larger amount of radiation heat transfer. By using a user-defined wall function that implements an empirical formula for the convection coefficient, a result similar to the SST model can be achieved with a considerably smaller amount of grid cells in the simulation model.

4.2. 3D model real radiator case

For the k-ε model with UDF, simulations with geometry of a real radiator model were performed with three different water supply temperatures and three different mass flow rates for each temperature. This model was studied in order to validate the UDF with the manufacturer experimental data from the Standard EN 442-2 test [19]. A comparison between experimental data and simulations is presented in Table 6.

The result from Table 6 shows that the simulation model fits the European standard quite well. The output heat transfer rate shows a maximum difference less than 0.5%.

Fig. 8a shows a comparison of the total heat transfer rate versus water mass flow rates for different supply/return water temperatures between the standard test and simulations. The simulated values (markers in Fig. 8) fit well to predicted values from Equations (1) and (2) (lines in Fig. 8). For each case three different mass flow rates were simulated. Fig. 8b shows the heat transfer rate versus the return water temperature, and the correspondence is good.

The surface temperature profile for a radiator with a water supply temperature of 55 °C and a water mass flow rate of 0.00442 kg/s is presented in Fig. 9. It can be seen that the highest temperatures are close to the inlet of the hot water and the lowest temperatures close to the outlet. An almost linear vertical temperature profile arises, but the temperature difference between the top and bottom decreases away from the inlet/outlet of the radiator.

4.3. Geometry simplification

The radiator geometry was simplified as a flat surface in order to decrease the computational time. Further the surface temperature of the radiator was set at a constant value according to the calculated logarithmic mean temperature in Equation (2). The heat transfer rate for this model versus manufacturer’s data [19] (Standard EN 442-2) is presented in Fig. 10. The first three cases were

Table 6: Results from experiment and simulations.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>75/55</td>
<td>19.5</td>
<td>0.00662</td>
<td>337.0</td>
<td>337.1</td>
</tr>
<tr>
<td>55/45</td>
<td>19.8</td>
<td>0.00442</td>
<td>179.6</td>
<td>178.8</td>
</tr>
<tr>
<td>45/35</td>
<td>19.5</td>
<td>0.00251</td>
<td>105.4</td>
<td>104.9</td>
</tr>
</tbody>
</table>
radiators with a height of 300 mm, and the last case has a radiator height of 590 mm. In all cases the water temperature from the radiator was 55/45 °C.

The results show that the used simplifications give a good agreement between the model and the Standard EN 442-2 test. For the 300 mm cases the maximum difference appears for the MP2 case with 0.2%. The difference is 1.4% for the MP1 radiator with 590 mm height, which is due to the constant (C₂) being adapted for the 300 mm radiator, but the error is still small.

4.4. Effects of the indoor climate

Simulations of the conditions in a room with dimensions according to the Standard EN 442-2 were performed. Results along a line 1 m ahead of the radiator from floor to ceiling showing the air velocity and temperature are presented in Fig. 11. Two cases were examined: a single-panel radiator (MP1) and a radiator with two panels (MP2). Heat transfer rates can be seen in Fig. 10: 180 W and 280 W, respectively.

Fig. 11a shows that a small increase in velocity appears for the MP2 case due to the increase in convective heat transfer. The velocity change is overall so small that some effects on the indoor comfort will not appear. Fig. 11b shows an increased temperature gradient for the MP2 case, which can have a larger effect on the indoor comfort.

5. Conclusions

The climate in buildings can be resolved correctly with CFD simulations using the k-ε model if user-defined wall functions are implemented on the radiator. The heat transfer coefficient predicted agrees well with the heat transfer coefficient obtained from the SST model, although the number of cells is reduced significantly. The proportion of heat transfer through convection and radiation is consistent with common knowledge.

The used model shows good agreement with measurements according to European Standard EN 442-2. Further simplifications incorporated in the simulations as a constant surface temperature, a flat radiator surface and describing the number of panels through a constant are possible to introduce without losing accuracy. A radiator with many panels or curvy geometry can then easily be simulated without the need to include all the geometrical details in the CFD model.

Although the working model is developed for a radiator height of 300 mm and a water supply temperature of 55 °C, the error of using it on a radiator with almost twice the height is small. In order to decrease the error, correct values of the constant can be determined for actual conditions through the described method.

CFD simulations of the indoor climate in buildings using the k-ε model must include user-defined wall functions on all surfaces with large temperature differences towards the air, for example radiators and windows.

Acknowledgement

We would like to thank VINNOVA for funding this work through the ATTRACT research project.

References


Paper III
CFD simulation and evaluation of different heating systems installed in low energy building located in sub-arctic climate

Daniel Risberg, Mattias Vesterlund, Lars Westerlund, Jan Dahl
Department of Engineering Science and Mathematics, Division of Energy Science, Energy Engineering, Luleå University of Technology, 971 87 Luleå, Sweden

Abstract

Computational Fluid Dynamics (CFD) simulations were used to study the indoor climate in a low energy building in northern Sweden. The building’s low heat requirement raise the prospect of using a relatively simple and inexpensive heating system to maintain an acceptable indoor environment, even in the face of extremely low outdoor temperature. To explore the viability of this approach, the indoor climate in the building was studied considering three different heating systems: a floor heating system, air heating through the ventilation system and an air heat pump installation with one fan coil unit. The floor heating system provided the most uniform operative temperature distribution and was the only heating system that fully satisfied the recommendations to achieve tolerable indoor climate set by the Swedish authorities. On the contrary, air heating and the air heat pump created a relatively uneven distribution of air velocities and temperatures, and none of them fulfills the specified recommendations. From the economic point of view, the air heat pump system was cheaper to be installed but produced a less pleasant indoor environment than the other investigated heating systems.

1. Introduction

The town of Kiruna is located in the very north of Sweden well above the Arctic Circle (Fig. 1) and is home to the world’s largest underground iron ore mine [1].

Mining activities in the vicinity of the town have caused extensive deformations of the ground, which are starting to affect the town itself as shown in Fig. 2. As a result, an extensive urban transformation of the area will be required in order to allow the continuation of mining operations [2].

Ideally, urban transformations should make the transformed region more climate-positive and energy efficient [3]. In order to achieve these goals in Kiruna, it will be important to reduce the energy utilization for newly built houses. The energy supplied to buildings is used to satisfy heating, hotwater and electricity demands. Notably, around 60% of the total energy supplied to the average Swedish house is used for room heating [4]. This value can however be greatly reduced by building low energy houses [5].

Nine passive houses located in Sweden has been investigated and compared with conventional buildings. The specific annual energy use for heating was in line with the predictions at a value of around 21 kWh/m² floor area and year [6]. Swedish building regulations state a maximum allowed value of 55 kWh/m² floor area and year in the same region [7].

Due to its northerly location, Kiruna has a sub-arctic climate with winter temperatures that often drops below −30 °C [8]. Consequently, the amount of energy spent on room heating in the average house within the town is significantly greater than the Swedish average, meaning that there is a considerable margin for reducing the town’s overall energy consumption by constructing energy-efficient housing.

The low energy building concept is based on the use of well-insulated envelopes made from components with low U-values and with relatively small quadruple-glazed windows. This results in reduced heating requirements, energy utilization, higher internal surface temperatures and lower air velocities than those found in conventional Swedish houses.

Several heating alternatives are available for new homes, as example radiators, floor- and air heating systems and point source units like a stove or fan coil. Each one of them has different installation cost. Radiators and floor heating systems with district heating is particularly costly to install, in fact when constructing a house these systems are around four times higher than the cost for an air heat pump system with single fan coil unit. Table 1 shows which heating source that is coherent with each different heating system.
been published. The research work has been focused on the
perform detailed investigations of the indoor climate within the
[9], but for sake of brevity only three of them are considered here.
concept has been studied with several different heating systems
Swedish building regulations. Actually the low energy building
good thermal indoor climate. The criteria were set according to the
heat pump system with one fan coil unit in a low energy building
air heating system with district heating was compared with an air
this period.
humidity of the heated indoor air may be as low as 10% RH during
the relative humidity to a lesser extent. The humidity of the out-
perature and velocity of the air and the incident radiation, as well as

<table>
<thead>
<tr>
<th>Nomenclature</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>C</td>
<td>cost (Euro)</td>
</tr>
<tr>
<td>c</td>
<td>cost (Euro/kWh)</td>
</tr>
<tr>
<td>E</td>
<td>energy demand (kWh)</td>
</tr>
<tr>
<td>DR</td>
<td>draught rating (%)</td>
</tr>
<tr>
<td>lw</td>
<td>turbulent intensity (%)</td>
</tr>
<tr>
<td>M.R.T</td>
<td>mean radiation temperature (T\text{a})</td>
</tr>
<tr>
<td>Nr</td>
<td>number (pcs.)</td>
</tr>
<tr>
<td>Ta</td>
<td>ambient air temperature (°C)</td>
</tr>
<tr>
<td>TR</td>
<td>operative temperature (°C)</td>
</tr>
<tr>
<td>v</td>
<td>velocity (m/s)</td>
</tr>
<tr>
<td>σ</td>
<td>Stefan Boltzmann constant (kg/(s² K⁴))</td>
</tr>
</tbody>
</table>

The indoor thermal climate is primarily affected by the tem-
perature and velocity of the air and the incident radiation, as well as
the relative humidity to a lesser extent. The humidity of the out-
door air is very low in Kiruna during the winter, and so the relative
humidity of the heated indoor air may be as low as 10% RH during
this period.

In this paper the performance of a floor heating system and an
air heating system with district heating was compared with an air
heat pump system with one fan coil unit in a low energy building
concept. These were evaluated and compared, and an analysis of
their operation was performed to assess their ability to satisfy a
good thermal indoor climate. The criteria were set according to the
Swedish building regulations. Actually the low energy building
concept has been studied with several different heating systems
[9], but for sake of brevity only three of them are considered here.
CFD (Computational Fluid Dynamics) simulations were used to
perform detailed investigations of the indoor climate within the
complete conditioned volume of the house.

Different studies concerning CFD simulation in buildings have
been published. The research work has been focused on the
computational setup for ventilation in a plane of a room [10].
Different k-ε turbulence models for 3D geometry were investi-
gated in Ref. [11]. A review of different turbulence models used in
buildings was done in Ref. [12]. In a single room installation,
 altered heating systems have been modeled and the indoor
climate studied [13], simulation of the indoor climate in an office
room with cooling ceiling has also been performed [14]. For the
whole building a passive house in Lindas in the south of Sweden
with an air heating system has been investigated with CFD tech-
nique, air velocities and temperatures in the building were investigated [15,16].

More specifically, the aims of this work were to:
  • Use CFD models in order to evaluate temperature and velocity
    field within an entire building.
  • Investigate how the indoor climate is affected by using different
    heating systems.
  • Evaluate the economical aspect of the different heating systems.
  • Make conclusions about the investigated heating systems and
    rank them according to different perspectives.
  • Evaluate the possibility to use district heating as a heating
    source for low energy buildings.

2. Method

2.1. Simulated building and boundary conditions

The investigated building, shown in Fig. 3, is a single family
house with three bedrooms (floor area of 98 m²) equipped with a
balanced ventilation system featuring a heat exchanger unit for
recovering heat from the exhaust air (the temperature efficiency is
70%). The building is oriented so that the entrance door opens to
the south. Window dimension is 1 m and the house has 31 win-
dows in total. The overall heat transfer coefficients (U-values) for
each structural element are presented in Table 2. Boundary setups
were assigned to the different building surfaces according to their
calculated U-values. The emissivity for the surfaces was set to 0.9
[17] for all building elements but the windows, for which a value of
0.84 [18] was used (Table 2). An annual heat demand of 8200 kWh
was estimated from the building specifications, and the annual hot
water production was set to 4500 kWh [19]. The peak heat supply
was estimated to be 2.6 kW with an indoor temperature of +20 °C
and an outdoor temperature of −30 °C. The time constant was
calculated from building specifications and typical furniture setup
and was found to be 37 h.

2.2. Swedish building regulations

Swedish regulations state that buildings should be designed so
that a satisfying indoor climate is obtained. A set of general rec-
ommendations are issued in order to set limits for achieving a
tolerable indoor thermal climate and the following criteria should
be fulfilled within the building occupied zone [7], these issues can
be omitted if the building constructor proves that a satisfying in-
door climate will be achieved.

1. Operative temperature (T\text{op}) not lower than 18 °C.
2. Maximum difference of 5 °C in each room (within the occupied
   zone both horizontal and vertical).
3. Floor temperature in the range of 16–26 °C.
4. Air velocity not higher than 0.15 m/s during the heating period
   and 0.25 m/s during the rest of the time.

Fig. 1. Location of the town of Kiruna, Sweden.
one 0.1 m above floor level and the other 2.0 m above floor level; and vertical boundaries that are 0.6 m distant from the exterior walls or other external limits, or 1.0 m from windows and doors [7].

The criteria listed above were used as evaluation parameters in order to assess the performance of the different heating systems considered.

### 2.3. Analysis parameters

Air temperature should not be the only parameter used to define the thermal indoor climate, but indeed it is the most commonly used. An indoor climate can be described as the aggregated experience for each individual person, affected by different variables such as air temperature, temperature of surrounding surfaces, relative humidity and air velocities [20]. In this paper the operative temperature (Eq. (1)) is calculated, which includes the ambient air

**Table 1** Interaction between energy supply systems and energy distribution systems.

<table>
<thead>
<tr>
<th>Energy distribution systems</th>
<th>Energy supply systems</th>
<th>District heating</th>
<th>Electrical boiler</th>
<th>Heat pump</th>
<th>Biofuel boiler</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water based</td>
<td>Floor heating</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td></td>
<td>Radiator</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td></td>
<td>Air heating</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td></td>
<td>Fan coil</td>
<td>X</td>
<td>X</td>
<td>✓</td>
<td>X</td>
</tr>
<tr>
<td></td>
<td>Stove</td>
<td>X</td>
<td>X</td>
<td>✓</td>
<td>✓</td>
</tr>
</tbody>
</table>

**Table 2** Data for building elements.

<table>
<thead>
<tr>
<th>Structural element</th>
<th>Area [m²]</th>
<th>U-value [W/m², K]</th>
<th>Emissivity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ceiling</td>
<td>96.0</td>
<td>0.085</td>
<td>0.9</td>
</tr>
<tr>
<td>Wall</td>
<td>87.1</td>
<td>0.100</td>
<td>0.9</td>
</tr>
<tr>
<td>Window</td>
<td>11.0</td>
<td>0.850</td>
<td>0.84</td>
</tr>
<tr>
<td>Entrance door</td>
<td>2.1</td>
<td>0.900</td>
<td>0.9</td>
</tr>
<tr>
<td>Floor</td>
<td>96.0</td>
<td>0.150</td>
<td>0.9</td>
</tr>
</tbody>
</table>

Fig. 2. Predicted deformation zones.

Fig. 3. Floorplan of investigated building specifying the location of the supply/exhaust ventilation devices, the air heat pump and the investigated vertical temperature gradient.
temperature \( T_\text{op} \), air velocity \( v \) and the mean radiation temperature (Eq. (2)) that is as a function of the radiation intensity \( I_{\text{rad}} \) from each surface. Due to relatively low air velocities in buildings, the operative temperature is a useful parameter in order to describe the indoor climate, since it includes the parameters that affect mostly the climate from a human perspective.

\[
T_\text{op} = \left\{ \begin{array}{ll}
\frac{M_\text{RT} + T_\text{ao} \sqrt{10v}}{(M_\text{RT} + T_\text{ao})/2} & (v \geq 0.1 \text{ m/s}) \\
(M_\text{RT} + T_\text{ao})/2 & (v < 0.1 \text{ m/s})
\end{array} \right. 
\tag{1}
\]

\[
M_\text{RT} = \left( \frac{\text{heat} \cdot c_p}{\sigma} \right)^{0.25}
\tag{2}
\]

In the definition of operative temperature the air velocity is one of variables, and a change at low velocities (but greater than 0.1 m/s) affects the operative temperature more than a variation at high velocities. In this study low velocities were expected and another way of determining the quality of the indoor climate is to verify the appearance of draught. Draught is not stated as a regulation to be avoided, but it is an important factor for indoor climate experience. The part of the human body that is most sensitive to draught is the face and a sitting person experiences draught more than a walking one. A parameter called Draught Rating (Eq. (3)) was introduced by Fanger at al [21], as an empirical expression based on ambient air temperature, air velocity and turbulence intensity \( I_\text{t} \). The result is meant as a statistic parameter that describes the share of a population that will be dissatisfied by draught. When the DR value is over 15% (i.e. 15% of a population), the indoor climate can be seen as unsuitable [22].

\[
\text{DR} = (34 - T_\text{ao})(v - 0.05)^{0.6} (0.37 I_\text{t} + 3.14)
\tag{3}
\]

Thus, Draught Rating can be considered as a useful parameter for a more advanced evaluation of the indoor climate.

### 2.4. Numerical setup

All the presented simulations were conducted using ANSYS CFX 15.0, a software package for numerically solving partial differential equations (PDEs) using a 3D-finite volume method. The governing equations that are solved in the simulations are those related to conservation of mass, momentum and energy. The calculations were performed using a second order upwind discretization scheme in which the normalized Root Mean Square (RMS) residuals converge to a level of \( 10^{-5} \). The simulations were performed under the assumption of steady state conditions, which applies to all the studied systems.

The turbulence model used in the simulations was the standard \( k-\epsilon \) model; a scalable wall treatment was used to obtain the temperature and velocity profiles in the near-wall region. The \( k-\epsilon \) model is a two equation model: \( k \) is the transport equation for the turbulent kinetic energy and \( \epsilon \) is the eddy dissipation [23]. The influence of buoyancy was predicted using the Boussinesq approximation. The radiation between different building surfaces was evaluated using the P=1 thermal radiation model [24]. All the simulations were run until a steady-state solution was achieved for a given set of outdoor conditions. A grid consisting of elements with a length of 0.1 m was selected based on the results of previous studies [25] using Roaches GCi method and Richardson extrapolation [26]. Inflation layers were used to achieve a better resolution at the near-wall regions, bringing the total number of elements required to simulate the entire building up to approximately 1.7 million.

### 2.5. Heating system setup

The three different distribution systems chosen for the simulations were: a floor heating system, an air heating system, and an air heat pump system. A schematic sketch of the different heating systems is shown in Fig. 4. The floor heating system is installed in each room of the house, the air heating system uses the floor air devices in the rooms as sources of supply air, and one air heat pump was installed above the entrance door and operates as a concentrated heat source (see Fig. 3). The air flow through the condenser unit was set to 0.164 kg/s, which can be seen as a standard setup for a typical air heat pump. The internal air was calculated to be heated up to 35 °C when the outdoor temperature became –30 °C. It should be noted that the simulations assume that all the internal doors in the house are open (except for the one leading to the bathroom).

The ventilation air flow was set according to Swedish building regulations [7]. Air supply devices are located in the living room and the bedrooms, with exhaust devices in the bathroom, kitchen, laundry and the storage space (Fig. 3). The supply and exhaust air flows are presented in Table 3; the total simulated ventilation air flow is 34 l/s. Normally the supply air temperature for air heating system is in the range between 40 and 50 °C [27], so a temperature of 45 °C is selected. In order to fulfill the heat demand, ventilation flows have to be adjusted according to Table 3. To be consistent with the conservation of mass, the air flows (supply and exhaust) were balanced in the simulations. For normal constructions the supply air flow is set to 50% of the exhaust air flow in order to create a negative pressure inside the building compared to the environment.

### 3. Results

#### 3.1. Operative temperature gradients

Hot air has a lower density, so warmer air will rise towards the ceiling. Due to continuity, air that is cooled by contact with relatively cold surfaces will descend towards the floor. These processes create vertical temperature gradients in each room and their steepness depends on the nature of the heating system used. Convective heat supply systems produce more pronounced vertical temperature gradients whereas those that rely on radiation for heat transfer result in smaller gradients. In addition, low energy buildings with modest heat losses produce less pronounced gradients than ordinary houses.

To evaluate the vertical operative temperature distribution obtained with the different heating systems considered in this work, calculations along a vertical line located 1 m from the window in the living room was performed (Fig. 3). This location was selected because it was considered likely to accurately illustrate a typical vertical operative temperature distribution produced by the studied heating system.

Fig. 5 shows the operative air temperature gradient for an outdoor air temperature of –30 °C. In the occupied zone (from 0.1 to 2.0 m above the floor) the floor heating system produced an almost uniform temperature distribution. The air heating system produced an operative temperature gradient of 1.2 °C and the air heat pump with the condenser located above the entrance door produced a temperature difference of 1.4 °C. In the air heat pump simulation, the highest temperature occurs at a height of 1.7 m above the floor. This is due to the significant movement of the indoor air caused by the condenser fan (Fig. 5).

To estimate the vertical operative temperature gradient throughout the house, two horizontal planes were created, one located 0.1 m above the floor and the other 2.0 m above the floor,
where the lowest and highest temperature was expected to be found. The highest temperature difference for these two planes was used as an indicator of the vertical temperature gradient for the overall house (in fact, if the gradient for the complete building is within the stated temperature range, then the gradient for each room is also fulfilled).

It was found that the operative temperature differences for the floor heating system are lower than 0.4 °C and there is little variation in this value throughout the building. A maximum difference of 4.3 °C is observed for the air heat pump system. Regarding the air heating system, the maximum difference is 20 °C in the vicinity of the air inlet device in bedroom 1, otherwise the temperature difference is lower than 3.8 °C. By this it is shown that the floor heating system and the air heat pump system fulfill the regulation stated by the authorities, but the air heating system does not. It can also be established that the air supply device in bedroom 1 has an inappropriate location when it is placed inside the occupied zone.

3.2. Floor temperature

As previously stated, the floor temperature should be in the range of 16–26 °C, in Table 4 the values of floor temperatures for the different heating systems is shown.

All systems are within the specified temperature levels and therefore fulfill the criterion. In all three cases the lowest temperature was found close to the entrance door, outside the occupied zone. For the air heating system, regions with cold spots are found around the inlet devices, which are also outside the occupied zone except for the device in bedroom 1.

3.3. Operative temperature, air velocity

To find the lowest operative temperature of the indoor air a horizontal plane at a height of 0.1 m is used for verifying if the heating system fulfills the stated criteria. In order to find the highest air velocity within the occupied zone different plane levels were tested and evaluated, and it is found that a horizontal plane in the range between 1.95 and 2 m is the most representative. Fig. 6 to 8 shows the simulated operative temperature and indoor air velocities for the entire house at the specified planes. The minimum level (blue color) in the left contour plot and the maximum level (red color) in the right contour plot are used to indicate values that are outside the range specified by the criteria.

3.3.1. Floor heating system

Fig. 6a shows that the differences in temperature over the entire house are very small when using floor heating as the heat source. Fig. 4 shows the results for the different heating systems.
distribution. In this case, the heat is supplied via a large surface the temperature of which is almost similar to that of the air, so the resulting temperature distribution is smooth. Fig. 6b shows that the air velocity field generated from floor heating system is characterized by small gradients and generally low air velocities.

The simulation shows that all values of the operative temperature and air velocity are within specified ranges when using a floor heating system.

### 3.3.3. Air heating system

The air heating system gives a uniform horizontal operative temperature distribution, although some warmer and colder regions are found close to the air inlets (Fig. 7a). The bathroom is colder than the rest of the house because it is only heated via the transfer of air from the hall. The temperature difference between the supply air and the indoor air will cause a column of air rising towards the ceiling quite close (less than 1 m) to the device. Air velocities that exceed the stated values were found mainly in the vicinity of air inlets and door openings.

The simulation showed that the operative temperature is within the boundaries when using an air heating system. The air velocity exceeds the stated value at several locations within the occupied zone. As a result of this, the air supply devices should be placed outside the occupied zone when planning a house with an air heating system. In particular, air transfer devices should be installed above the occupied zone between the rooms in order to lower the air velocities within the occupied zone.

### 3.3.3. Air heat pump system

Simulations were conducted with an air heat pump unit located in the entrance hall as shown in Fig. 3 (in practice heat pumps are usually installed above the entrance door). This layout yields the lowest variation in operative temperature compared to other simulated locations of the air heat pump, which is positive for the thermal climate.

In this case, the operative temperature show the most unevenly distribution among the three considered systems (Fig. 5a). Moreover, the internal air velocities exceed the upper limit of 0.15 m/s mainly in the hallways and in the living room.

The simulation showed that the operative temperature of the air heat pump system is within the boundaries set by the authorities. The air velocity exceeds the recommended maximum level at several locations within the occupied zone. It is hard to find a location for an air heat pump that fulfills the stated criteria.

### 3.4. Draught rating

In Fig. 9 the Draught rating is shown for a horizontal plane at 1 m above the floor, which was chosen as representative of the height for the face of seated person [28]. With the floor heating system the effect of draught is small and values above 15% appears only outside the occupied zone (Fig. 9a). With the air heating system high values are shown close to the ventilation supply devices, where the velocity is high (Fig. 9b). Finally, with the air heat pump system large areas inside the living room show values above 15% (Fig. 9c).

### 3.5. Economic evaluation

In order to enlarge the perspective about the criteria according to which the considered heating system should be ranked, a techno-economic evaluation was performed. The energy supply for the air heat pump is electricity and for the other systems the hot water from district heating was chosen.

In Sweden, most buildings are heated by a local district heating system. On the secondary circuit (i.e. in the building), the heat is typically distributed using radiators or floor heating systems. Kiruna has a well-developed district heating system to which most of town buildings are connected; it supplies 90% of the town’s total heat requirement. This system primarily generates heat by burning a mixture of fuels, including waste, biomass and oil, together with some electric heating. The district heating system is considered to be a sustainable and environmentally friendly heat production technology. Air heat pumps consume electricity, which is an energy source that ideally should be conserved. The cost of the energy consumed by the considered systems is an important factor when comparing their convenience.

The initial cost of connecting a house to the district heating network in Kiruna is around 4400 € and, based on current tariffs, the heat supplied by the system costs consumers 0.30 €/kWh. This is approximately equal to the national average cost of heat supplied by district heating systems. The system installation cost is 4495 € for floor heating [29] and 910 € for air heating.

<table>
<thead>
<tr>
<th>Floor temperature</th>
<th>Operative room temperature</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimum [°C]</td>
<td>Average [°C]</td>
</tr>
<tr>
<td>Floor heating</td>
<td>17.0</td>
</tr>
<tr>
<td>Air heating</td>
<td>16.1</td>
</tr>
<tr>
<td>Air heat pump</td>
<td>17.8</td>
</tr>
</tbody>
</table>

In Fig. 6 the a) operative temperature distribution at 0.1 m, b) air velocity field at 2.0 m.
Fig. 7. Air heating system, a) operative temperature distribution at 0.1 m, b) air velocity field at 1.95 m.

Fig. 8. Air heat pump system, a) operative temperature distribution at 0.1 m, b) air velocity field at 2.0 m.

Fig. 9. Draught rating distribution for a) floor heating system, b) air heating system, c) air heat pump system.
cost compared to a traditional ventilation system). The operation lifetime for both systems is assumed to be 30 year.

The cost of purchasing and installing an air heat pump is around 2800 € (2100 € for materials and 700 € for the installation) [30]. Based on average daily temperature measurements for the period between 2008 and 2011 [8] and data from an independent testing organization [31], which was testing 17 different air heat pumps for three different locations in Sweden, an average annual COP of 2.3 was calculated for an air heat pump located in Kiruna. At the current electricity price of 0.12 €/kWh, this is equivalent to a heat cost of 0.052 €/kWh. The air heat pump was assumed to have an operational lifetime of 15 years and therefore would need to be re-installed once during a 30 year period.

In addition to room heating, the energy consumed for production of hot water must be considered. This is normally done by using electricity in houses that have an air heat pump. The studied house has three bedrooms and is assumed to be occupied by a typical Swedish household of two adults and two children. On average, this household would require 4500 kWh/year for the production of hot water [19]. Cumulative costs for the household using the systems considered in this work over a period of 30 years were calculated according to Eq. (4). The equation includes the initial cost of installing the heating system ($C_{\text{inst}}$), how many times it has to be replaced ($N_{\text{units}}$), the operational lifetime of the system ($t_{\text{op}}$), any applicable connection fees ($C_{\text{connection}}$), as well as the unit cost of energy for room heating ($C_{\text{heat}}$) and hot water production ($C_{\text{hotwater}}$). The setup and result using Eq. (4) are shown in Table 5.

$$\sum_{t_{\text{op}}} C_{\text{system}} = N_{\text{units}} C_{\text{inst}} + C_{\text{connection}} + E_{\text{hotwater}} C_{\text{hotwater}} + E_{\text{heat}} C_{\text{heat}}$$

3.5.1. District heating price level

As shown in Table 5 the air heat pump system is the least expensive over a 30 year period and the most expensive heating system is floor heating. In order to make the air and floor heating systems attractive from the economic point of view, their cost has to match with the cost for an air heat pump system. In Fig. 10, the yearly cost for the different systems can be seen.

The unit cost of heating for both air and floor heating systems is allowed to be higher than that of the air heat pump system, because of the higher unit cost for making hot water with electricity. The district heating price for the air and floor heating systems has to be reduced by 23 and 33%, respectively, to match the total cost of an air heat pump system.

### Table 5

<table>
<thead>
<tr>
<th>System</th>
<th>System installation</th>
<th>Con. + inc. for</th>
<th>COP</th>
<th>Heating</th>
<th>Hot water</th>
<th>Technical lifetime</th>
<th>Total cost</th>
<th>Yearly cost</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>[€]</td>
<td>[€]</td>
<td></td>
<td>[kWh/kWh]</td>
<td>[€]</td>
<td>[Years]</td>
<td>[€]</td>
<td>[€/year]</td>
</tr>
<tr>
<td>Air heating</td>
<td>2800</td>
<td>4400</td>
<td>0.1</td>
<td>0.052</td>
<td>0.12</td>
<td>15</td>
<td>34,524</td>
<td>1151</td>
</tr>
<tr>
<td>Floor heating</td>
<td>4495</td>
<td>4400</td>
<td>0.1</td>
<td>0.052</td>
<td>0.12</td>
<td>30</td>
<td>46,995</td>
<td>1567</td>
</tr>
</tbody>
</table>

3.5.2. District heating compared with electricity price level

In order to have a break even with the air heat pump the unit cost of energy for an air heating system should be 64% of the electricity price and only 56% for the floor heating system in order to have the same total cost as the air heat pump system for a period of 30 year. In these calculations the development of market prices was not taken into account, but if electricity and district heating prices maintain their ratio the results are still valid.

When the price variation of electricity is taken into account, limit values for the district heating price can be found below which floor and air heating system are more convenient. Considering any given electricity price, the set of the corresponding limit values forms a curve for the considered system (floor or air heating), as shown in Fig. 11. The current price for district heating is 83% of the electricity price and this makes the air heat pump a more profitable heating system to install from an economical perspective. In order to make air heating or floor heating system more appealing with current electricity price the cost for district heating has to decrease along the dotted line in Fig. 11.

As can be seen in Fig. 11, with a low electricity price the installation and connection fee represent the dominant part of the total cost, which are an advantage for the air heating system and a drawback for the floor heating system. With an increased electricity price the energy cost will be the dominant part of the total cost during 30 years, and the limit ratio between the district heating and electricity prices tends to be somewhat higher than 60% as the fixed part of the total costs become relatively unimportant.

### 4. Discussion

The urban transformation of Kiruna will necessarily involve the construction of many new buildings. If the builders and planners only consider the total cost of construction and operation when...
selecting the heating systems for these homes, they may install air heat pumps or alternative heating systems rather than using the district heating service. However, by using the district heating service it becomes possible to fit water-based heating systems in the homes, which provide the best indoor thermal environment as discussed in the previous sections. In public buildings and industrial facilities, the use of the district heating systems is practically more convenient; to minimize network losses and installation costs, such buildings should be grouped together when rebuilding the town.

In the current situation there is only a partial cooperation about the heat transfer between Kiruna’s district heating network and the local mining industry, which produces large quantities of waste heat. As a result, the margin for reducing the cost for heat generation in the town is likely to be considerable by expanding this cooperation. This would further increase the attractiveness of using district heating technology in new housing.

In Table 6 a performance summary for the considered heating systems is shown according to the stated recommendations set by the Swedish authorities, indicating with an X the advices that are not fulfilled.

The floor heating system generates an even distributed temperature field with low air movement and near-ideal operative temperatures. Local or sub-local heat distribution systems (i.e. air heating and air heat pump) generate larger differences in temperatures and air velocities within the building, the internal air velocities exceed the maximum advised values specified in the Swedish building code. The air heating system exceeds the temperature difference (as marked with an asterisk in Table 6) in bedroom 1, where the air inlet device is placed within the occupied zone. When planning for a house with air heating it is important to place the air inlet devices outside the occupied zone and air transfer devices installed between rooms in order to reduce the air velocity below the limits of the stated criteria.

The larger temperature gradients observed when using air heating and air heat pump systems occur because these heat sources increase the strength of natural convection, and it should be noted that if some doors were closed even higher temperature differences would be created.

Air heat pump manufacturers suggest that the size of the heat pump should be calculated so that it can cover 100% of the maximum heat requirement, even in sub-arctic climate. This was the sizing criterion that was adopted for the simulation of the air heat pump system.

Besides the discussed parameters, the quality of indoor climate includes also the absence of odors and polluting substances, lighting and low noise levels which are not included in this work. Maintenance and availability are other factors that must be included in the final decision about the heating system to be installed.

5. Conclusions

The results show that only floor heating system fulfills the recommendations set by the authorities to achieve a tolerable thermal indoor climate in a low energy building located in a sub-Arctic environment.

The investigated heating systems can be ranked as follows according to the quality of the indoor thermal environment, starting with the best:

- Floor heating system.
- Air heating system.
- Air heat pump system.

A local concentrated heat source such as in air heating and air heat pump systems provides a poorer indoor climate in terms of operative temperature variations, air velocity, and Draught rating. The floor heating system creates almost negligible temperature gradients with very low air motions.

From an economical point of view the air heat pump is the cheapest alternative for heating buildings, over the studied period district heating with air and floor heating system is 22% respectively 33% more expensive than the air heat pump. To make district heating an economically attractive heating source for households a dramatic increase in the cost of electricity or a significant reduction in price for district heating is needed.

Acknowledgments

We would like to thank FORMAS, VINNOVA and our academic partners at Luleå University of Technology who made this research possible.

<table>
<thead>
<tr>
<th>Summary</th>
<th>$T_{\text{in}} &gt; 18^\circ \text{C}$</th>
<th>$dT &lt; 5^\circ \text{C}$</th>
<th>$T_{\text{floor}} &gt; 16^\circ \text{C}$</th>
<th>$V_{\text{avg}} &lt; 0.15 \text{ m/s}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Floor heating</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Air heating</td>
<td>✓</td>
<td>✓*</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Air heat pump</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>×</td>
</tr>
</tbody>
</table>
Paper IV
Investigation of thermal indoor climate for a passive house in a sub-Arctic region using computational fluid dynamics

Daniel Risberg, Mikael Risberg and Lars Westerlund

Abstract
There is currently an increasing trend in Europe to build passive houses. In order to reduce the cost of installation, an air-heating system may be an interesting alternative. Heat supplied through ventilation ducts located at the ceiling was studied with computational fluid dynamics technique. The purpose was to illustrate the thermal indoor climate of the building. To validate the performed simulations, measurements were carried out in several rooms of the building. Furthermore, this study investigated if a designed passive house located above the Arctic Circle could fulfil heat requirements for a Swedish passive house standard. Our results show a heat loss factor of 18.8 W/m² floor area and an annual specific energy use of 67.9 kWh/m² floor area, would fulfils the criteria. Validation of simulations through measurements shows good agreement with simulations if the thermal inertia of the building was considered. Calculation of heat losses from a building with a backward weighted moving average outdoor temperature produced correct prediction of the heat losses. To describe the indoor thermal climate correctly, the entire volume needs to be considered, not only one point, which normally is obtained with building simulation software. The supply airflow must carefully be considered to fulfil a good indoor climate.

Keywords
CFD simulations, Indoor climate, Passive houses, sub-Arctic climate, Validation

Accepted: 21 December 2017

Introduction
A passive house is a well-known phenomenon in the building construction industry. There is an increasing trend in central Europe, especially in Germany, Austria and Switzerland to build passive houses.1 Also in Sweden, the number of passive houses has increased in recent years.2 The passive houses that are built in Sweden are mainly located in the southern region due to its warmer climate compared to the northern part of Sweden. The cold climate in this region increases the demand on the building’s envelope surfaces. The northernmost built passive house is located in the Swedish town of Umeå. In 2013, the construction company NCC decided to build a passive house in the town of Kiruna located above the Arctic Circle and around 600 km north of Umeå.3 For this region, the yearly mean outdoor temperature is −1.2°C and the minimum outdoor temperature is colder than −30°C.4

Passive houses were earlier studied by Badescu and Sicre5 with 1D time-dependent conduction heat-transfer to determine the heat load of a building located in Germany. The study showed that solar heating can provide heat for the main part of the year. The town Kiruna that is located above the Arctic

Division of Energy Science, Luleå University of Technology, Luleå, Sweden

Corresponding author:
Daniel Risberg, Luleå University of Technology, SE-971 87 Luleå, Sweden.
Email: daniel.risberg@ltu.se
Circle has negligible contribution from the sun over a long period of time.

A study of the indoor climate for nine different passive houses located in Linköping, in the south of Sweden was performed by Rohdin et al. The work was based on both simulations and post-occupancy evaluation. The result showed that the indoor climate was in overall good but from the post-occupancy evaluation, complains of cold floors and too high temperatures during the summer was more common.

A study by Sartori and Hestnes compared the embodied energy and operating energy during its lifetime for different types of buildings. They showed that the main part of the energy use was during the operating time. For a passive house, their study showed a minor increase in embodied energy and a substantial decrease in operating energy compared to a conventional building.

A higher cost of thicker walls in order to reduce the thermal losses from the building’s envelope surfaces would entail a reduction of the investment cost for the heating system. An air-distributed heating system is often used instead of an expansive hydronic heating system. This is a less studied heating method compared to radiators and underfloor heating. These systems have been demonstrated to give a good thermal indoor climate, due to its high contribution of thermal radiation from the heat source. Feist et al. investigated if a good thermal indoor climate was achievable according to EN ISO 7730:2005 standard for an air heating system. The study concerned a single room with heated supply air and showed a good thermal indoor climate according to EN ISO 7730:2005 standard.

A study of the indoor climate for nine different passive houses located in Linköping, in the south of Sweden was performed by Rohdin et al. The work was based on both simulations and post-occupancy evaluation. The result showed that the indoor climate was in overall good but from the post-occupancy evaluation, complains of cold floors and too high temperatures during the summer was more common compared to traditional buildings.

Studies using CFD simulations for indoor building applications have been more common. Several studies focused on a single room with different heating systems like radiators and underfloor heating and with different ventilation setups. Comparisons between different heating systems like air heating with supply displacement ventilation ducts, radiators and underfloor heating have also been made. Simulation of cooling systems using air is fairly well documented. A study by Awbi has shown that convective heat transfer coefficients and radiation between surfaces could have a major impact on the thermal indoor climate.

Studies of passive houses with CFD simulations had been previously performed by Bajc et al., which investigated effects of Trombe walls on energy demand and indoor temperatures. Fokaides et al. investigated the indoor climate in subtropical climate and Karlsson and Moshfegh investigated different control strategies and boundary condition effects on the indoor climate for a low energy building. Previous work using CFD to study effects of supply devices at the ceiling for an air heating system lacked appropriate experimental validation. The temperature gradient in two-storey buildings and in a house with heat distribution in only some rooms has not yet been studied. The main upside by using CFD simulation, compared to other modelling techniques is that the CFD simulation can predict the indoor climate in the whole building. Other building simulation software do not consider vertical temperature gradients and often only describe the indoor climate as a point value in each room.

The increased thermal resistance of the building envelope surfaces reduces the thermal losses and in that way improves the thermal indoor climate. The main difference between different heating systems is the amount of radiation from the heat source. A radiator system with a single panel radiator has a smaller gradient compared to multi-panel radiators. For an air heating system, the amount of radiation is small, which should create an increased temperature gradient.

The aim of this paper was to find out if the thermal indoor climate could be simulated for an expected passive house with an air heating system. The heat was supplied through ventilation ducts located at the ceiling. In order to validate the CFD simulations, manual measurements were performed in several rooms of the building.

Building description

The building was constructed as a twin house with a living space of 140 m² for each apartment with
U-values for the envelope surfaces presented in Table 1. Both apartments have a solar cell system installed with a total area of 15 m² each and a green roof with sedum plants. In Figure 1, the building can be seen together with its geographical location. The heating and ventilation system was separately installed for each apartment and measurements were made in both apartments. In this paper, the focus was on the left apartment (see Figure 1).

In order to fulfil the Swedish requirements for a passive house located in Kiruna, the building must have an annual weighted specific energy use below 77 kWh/m² floor area and a maximum heat loss factor below 19 W/m² floor area. In order to accomplish that demand, not only the transmission losses had to be reduced compared to other passive houses in the south of Sweden. The hot water usage also had to be reduced and solar cells had to be installed. The geographical location above the Arctic Circle in a sub-Arctic region, this has made it very challenging to fulfil the demands for a passive house.

The building was heated with a low temperature district heating system (DHS). Inside the building, the district heating was connected to a heating coil in the ventilation system in order to heat up the supply air, see Figure 2. The ventilation system was installed with a rotational heat exchanger (HEX) without recirculation of exhaust air. In order to reduce the amount of hot water (HW) usage to the building, showers that recycle the water for each user were installed. The showers use electricity to reheat the water during showering. The reduction represents 80% of the total shower heat demand of the annual standard value. A schematic figure of the heating system is presented in Figure 2.

Swedish building requirements have a minimum limit on supply airflow of 1.26 m³/h per m² of floor area and a maximum supply ventilation air temperature of 52°C. The supply devices were located in bedrooms and living rooms. Exhaust air devices were placed in bathrooms, kitchen and the laundry room. In total, seven supply devices (three on the ground floor and four on the first floor) and four exhaust devices (three on the ground floor and one on the first floor). All devices were placed at the ceiling of each room. The supply devices were placed at a distance of 0.40 m from the inner walls. A circular shape with a diameter of 125 mm was used. The flow was distributed uniformly. The supply air temperature was varied and was set by the control system to fulfil the set point value of the indoor air temperature. The exhaust air devices were circular with a diameter of 100 mm and with a distance of 0.5 m from the inner walls.

During measurement periods, no occupants were living in the building. Therefore, no heat from occupants and a small amount of heat (114 W) from indoor apparatus were entered into the CFD simulation. For the annual case, the hot water usage had to be corrected for comparison. The indoor air temperature was too low during the coldest periods and consequently the supply airflow was increased on 25 November 2016. The annual energy balance presented was generated before the change.

Method

Continuous measurements of heat and electricity to the facility (A:0 in Figure 3) was performed to generate an energy balance (A:1) of the building in order to evaluate the specific energy use (A:2). The specific
annual electricity energy usage was calculated using equation (1) according to the passive house standard. The heat demand \((B:0)\) was calculated with basic equations (equations (2) to (5)), variable data from measurements and backward weighted moving average outdoor temperature (BWMAOT) (equation (6)). This was done for different BWMAOT in order to establish thermal inertia of the building \((B:1)\). The comparison between calculated and measured values gave an accurate value of the BWMAOT, to calculate the heat loss factor for the building \((B:2)\).

To evaluate the thermal indoor climate, CFD simulations were performed, with boundary conditions from basic equations \((C:0)\). The developed CFD model was validated against manual measurements of temperature and velocity inside the building \((C:1)\). Furthermore, the validated model was used to predict the thermal comfort for two different cases \((C:2)\), an outdoor temperature of \(-5^\circ\text{C}\) (outdoor temperature during manual measurements) and \(-25.7^\circ\text{C}\) (design outdoor winter temperature \(\text{DOWT}\) for the heating system\(^28\)) in Figure 3, a summary of the working process. The temperature gradients obtained in chosen rooms were also compared with earlier studies of thermal indoor climate in sub-Arctic climate.

**Passive house standard**

The building was investigated in order to see if the Swedish building requirements for a passive house were achieved. The annual energy usage was summed up from measurements for a whole year. To represent a normal year in Kiruna, the annual heating values were scaled according to the degree-day method. The electricity produced by the solar panels was measured as well as the total electricity used in the building. When the solar panel produced more electricity than was needed in the building, the surplus was transferred to the grid and presented as sold electricity. This amount cannot be included in specific energy calculations.

In order to calculate the amount of produced electricity that reduce the specific energy usages, measurement values on minute basis over the year were studied. According to the passive house standard, the different energy carriers from electricity \((E_{el})\), district heating \((E_{DH})\), cooling \((E_{cold})\) and other energy sources \((E_{Remain})\) are weighted according to equation (1).\(^{26}\)

\[
E_{\text{weighted}} = 2.5E_{el} + 0.8E_{DH} + 0.4E_{cold} + E_{Remain}
\]  

(1)

No occupants were living in the house, therefore the heat from electric appliances was low and did not benefit the heating system (household electricity is not included). Furthermore, no hot water was used during the measurement period. The hot water production had to be based on standard annual values.\(^{29}\) The showers recirculation of water reduced the total annual usage of hot water by 40% compared to the standard value of 20 kWh/m\(^2\) floor area. The balance of indoor air temperature for a passive house was set at \(17^\circ\text{C}\). The internal heat production and radiation from the sun would give a surplus, so the indoor air temperature was assumed to reach \(21^\circ\text{C}\). If the indoor air temperature (mean monthly value) was below \(17^\circ\text{C}\), the heat consumption was scaled up in the analysis to fulfil the requirements.

The second criterion that should be fulfilled for a passive house is a maximum heat loss at the design outdoor winter temperature less than 19 W/m\(^2\) K. The heat losses were calculated according to basic equations for an outdoor temperature of \(-25.7^\circ\text{C}\) and an indoor temperature of \(21^\circ\text{C}\). The time constant of the building was calculated to 7.3 days.
Thermal comfort

The thermal comfort was investigated with Fanger’s comfort model\textsuperscript{30} and was implemented in the simulation for investigated cases. The model calculates the number of dissatisfied people in percentage, based on the room temperature generated from both convection and radiation heating, relative humidity, air velocity, metabolic rate and clothing. The metabolic rate used was based on ASHRAE Standard\textsuperscript{31} with a value of 52.8 W/m\textsuperscript{2} surface area of the body. An adult human figure represents around 1.7 m\textsuperscript{2}. The clothing was set for winter indoor case with 1 Clo, which represents a heat resistance value of 0.155 m K/W. The water vapour pressure was set at a value of 467 Pa based on the measured values of indoor air temperature and relative humidity.

The EN ISO 7730\textsuperscript{11} defines three categories of thermal indoor climate (A, B and C) that are presented in Table 2. The thermal comfort categories are based on the Fanger comfort model (PPD), the PMV, the draft ratio (DR) and the local thermal discomfort (PD). The local thermal discomfort for our study was divided into temperature gradient, floor temperature and radiant asymmetry.

The Swedish building regulations\textsuperscript{32} describe the occupied zone as ‘enclosed by two horizontal levels, one 0.1 m above floor level and the other 2.0 m above floor level, a vertical level 0.6 m from exterior walls or other external limits, or 1.0 m by windows and doors’. In the occupied zone of each room, the regulations specify an operative temperature above 18°C, a maximum temperature gradient of 5°C in each room, a floor temperature in the range of 16-26°C and air velocities below 0.15 m/s during the winter season.

Installed measurement devices

The house was equipped with sensors for continuous measurements for verification of the building energy performance and thermal indoor climate. In total,
23 sensors were installed, where (T1–T10) in Figure 4 measures the temperature and relative humidity in different rooms with a sampling frequency of 60 s. These gauges were placed at the middle (floor/ceiling) of the wall. The ventilation system had five temperature sensors installed (T11–T15), three were mounted in the supply air channel and two on the exhaust airflow, before and after the heat exchanger (see Figure 2). The district heating used for the heating demand and hot water production was registered with energy meters from Kamstrup. The supply airflow (mass flow) was calculated from the measured temperature before/after heating coil and measured heat to the apparatus. The building electricity was divided into one electricity meter, which measured the total electricity supply to the building, and one electricity meter registered the supply to the car engine heater. Electricity meters were also installed for each shower and on each solar panel. In order to study if the passive house regulations were fulfilled, measurements were conducted during 2015 and the main part of 2016. The measurement period for validation of theoretical calculations was a 48-day period from 28 November 2016 to 15 January 2017.

**Manual measurements.** To evaluate the CFD model, manual measurements with a hot wire anemometer and temperature sensor were performed. In Figure 4, P1–P4 represents locations of different points (∼0.5 m distance) in a vertical line from floor to ceiling in order to obtain a temperature gradient. The lines were chosen in rooms where people spend most of their time and located in the occupied zone. Furthermore, rooms were chosen on both floor levels and with supply air or exhaust air (kitchen). P1 and P3:s position were chosen since the largest temperature gradient in these rooms were expected to be in front of windows, P2 and P4:s position were based on an expectation that the area is frequently occupied in the room. V1–V5 represents velocity measurement points; the points were 10 mm from the ceiling and in door space; the used point was 10 mm from the top of the doorframe. The locations were selected to obtain sufficiently high velocities, even if they are placed outside the occupied zone. The manual measurement devices were calibrated with a temperature error of ±0.5°C and for the velocities ±0.03 m/s and ±1% of the measurement value.

The results were carried out with an average value during 30 s measurements in every individual measurement point. The manual measurements were carried out on 14 December 2016.

**Simulation setup**

The simulations were carried out with the commercial software Ansys CFX 17.2, where the k-ε model was used together with user-defined wall functions for convective heat transfer coefficients described in Risberg et al.25 The coefficients were based on Awbi and Hatton33 and are presented in Table 3, where D is the hydraulic diameter and AT is the temperature gradient between the surface and the reference point. The buoyancy effect was predicted using the Boussinesq approximation. For radiation, the discrete transfer radiation model34 was used with an emissivity of 0.9 for opaque building surfaces and 0.83 for windows. The inner walls were simulated as solid walls to correct the predicted heat transfer between different rooms. The same assumption was used for the system of joists to predict heat transfer between different floor levels. The thermal conductivity used for these elements was 0.05 W/m K. The grid size of the inner building volume was specified to 0.1 m as maximum cell face length and with inflation layers against building surfaces according to Risberg et al.35 together with refinements close to ventilation air inlets. In work done by Risberg et al.,35 a grid convergence study was performed according to Roache.36 It shows small differences in temperature values for different grids and an uncertainty of maximum ±0.015 m/s for velocities.

The geometry of the simulation volume is presented in Figure 5. In total, the model had 3.3 million elements. For all governing equations and turbulence model equations, a convergence target of 1e-6 for the uncertainties of maximum

<table>
<thead>
<tr>
<th>Surface type</th>
<th>Convective heat transfer coefficient [W/m², K]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wall</td>
<td>( h_c = 1.823/D^{0.121} \Delta T^{0.293} )</td>
</tr>
<tr>
<td>Floor (cold surface facing upward)</td>
<td>( h_c = 0.704/D^{0.601} \Delta T^{0.133} )</td>
</tr>
<tr>
<td>Ceiling (cold surface facing downward)</td>
<td>( h_c = 2.175/D^{0.078} \Delta T^{0.308} )</td>
</tr>
</tbody>
</table>

During the measurement period, the temperature of the supply air temperature was continuously fluctuating between 21°C and 45°C. Due to the high thermal mass of the building, the inlet supply air temperature to each room was simplified with a constant inlet temperature based on the average temperature. The supply
airflow data to each room was set according to Table 4 and exhaust air mass flow according to Table 5. The boundary for exhaust air in the kitchen was set to pressure outlet in order to fulfill the mass balance. The air change rate was calculated to 0.94 ACH and the air was flowing between rooms through door openings or the gap under the closed door (bathrooms and laundry). The furniture was excluded since they only have a minor effect on the temperature distribution. Zhuang et al.37 compared three cases with different furniture layout, they all resulted in similar temperature gradients and differences were mainly caused by computer (heat source) and occupants location. The supplied electricity (114 W) in CFD simulations was assumed to provide heat to the building in the bathroom and the laundry. The showers would use some amount of electricity (around 15 W) continuously, most of the technical equipment were installed in the laundry like router and measurement base station (84 W).

In the simulation model, the heat losses were set as heat fluxes on every individual surface facing the outdoor environment according to the calculated transmission and infiltration losses. Outer walls, windows and so on were thereby not included in the simulation model. To describe the heat fluxes used in the steady-state CFD model, it was essential to determine a weighted value for the outdoor temperature. All variables in the calculation of the heat losses from the building were constant except the outdoor temperature. During the used measurement period, the contribution from the sun can be neglected (above Arctic Circle). A parametric study of number of days used in the backward weighted moving average outdoor temperature was performed to investigate if the heat losses could be predicted on an hourly basis with basic equations (2) to (5). The basic equations (2) to (5) were compared with measured data in order to verify the heat losses from transmission ($Q_{\text{trans}}$), infiltration ($Q_{\text{inf}}$) and ventilation ($Q_{\text{vent}}$). The thermal mass of the building has no impact on the thermal indoor climate for a steady-state case. When the outdoor air temperature changes over time, the thermal losses through climate envelope surfaces have a large impact on the thermal mass of the building. The stored heat in the building consequently has an effect on the heat demand.

$$\dot{Q}_{\text{trans}} = \sum UA_{\text{tot}} \Delta T_w$$  \hspace{1cm} (2)

$$\dot{Q}_{\text{inf}} = \dot{V}_{\text{infl}} c_p \rho \Delta T_w$$  \hspace{1cm} (3)

$$\dot{Q}_{\text{vent}} = \dot{V}_{\text{vent}} c_p (T_{14} - T_{12})$$  \hspace{1cm} (4)

$$\dot{Q}_{\text{tot}} = \dot{Q}_{\text{trans}} + \dot{Q}_{\text{inf}} + \dot{Q}_{\text{vent}}$$  \hspace{1cm} (5)

$$T_w = \frac{\sum_{n=1}^{n}(n-1) \cdot T_1}{\sum_{n=1}^{n}(n-1)}$$  \hspace{1cm} (6)

Table 5. Exhaust air mass flow.

<table>
<thead>
<tr>
<th>Room</th>
<th>Mass flow (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kitchen (ground floor)</td>
<td>Open</td>
</tr>
<tr>
<td>Bathroom 1 (ground floor)</td>
<td>0.015</td>
</tr>
<tr>
<td>Laundry (ground floor)</td>
<td>0.015</td>
</tr>
<tr>
<td>Bathroom 2 (first floor)</td>
<td>0.025</td>
</tr>
</tbody>
</table>

Table 4. Supplied air data.

<table>
<thead>
<tr>
<th>Room</th>
<th>Mass flow (kg/s)</th>
<th>Velocity (m/s)</th>
<th>Turbulent intensity (%)</th>
<th>Temperature (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Living room (ground floor), 2 devices</td>
<td>0.0150</td>
<td>0.35</td>
<td>5</td>
<td>34.7 (45.4)</td>
</tr>
<tr>
<td>Family room (first floor)</td>
<td>0.0085</td>
<td>0.39</td>
<td>5</td>
<td>34.7 (45.4)</td>
</tr>
<tr>
<td>Bedroom 1 (first floor)</td>
<td>0.0125</td>
<td>0.58</td>
<td>5</td>
<td>34.7 (45.4)</td>
</tr>
<tr>
<td>Bedroom 2 (first floor)</td>
<td>0.0125</td>
<td>0.58</td>
<td>5</td>
<td>34.7 (45.4)</td>
</tr>
<tr>
<td>Bedroom 3 (first floor)</td>
<td>0.0125</td>
<td>0.58</td>
<td>5</td>
<td>34.7 (45.4)</td>
</tr>
<tr>
<td>Bedroom 4 (ground floor)</td>
<td>0.0140</td>
<td>0.65</td>
<td>5</td>
<td>34.7 (45.4)</td>
</tr>
</tbody>
</table>

Figure 5. Geometry used for the computational fluid dynamics model.
For the ground floor, a constant soil temperature was used and for the other envelope surfaces a 10-day backward weighted moving average outdoor temperature was used. This value was subtracted from the mean measured indoor air temperature for each floor level for calculation of $D_{Tw}$. The temperature $T_{12}$ and $T_{14}$ location of equation (4), are presented in Figure 2. The ventilation losses were calculated with an estimated ventilation airflow ($\dot{V}$) and the temperature differences from measurements, where $c_p$ is the specific heat capacity and $\rho$ is the density. The infiltration flow rate ($\dot{V}_{\text{infl}}$) was prescribed as 0.12 m$^3$/h per m$^2$ of envelop area.

**Simulation case 1.** Manual measurements were performed in order to validate the CFD model. They were executed on the 14 December 2016 with an outdoor temperature of $-5^\circ$C. This day can be said to represent a normal winter day. Indoor air temperatures and air velocities were measured in different points (see Figure 4). During these measurements, the supply air temperature was on average $34.7^\circ$C, in order to cover the heat losses from the building. This value was used as inlet boundary condition for the supply air during simulations.

**Simulation case 2.** For the second case, the CFD model was used with an outdoor temperature of $-25.7^\circ$C. This case is a steady-state case with an outdoor temperature that is a low temperature even in Kiruna. The case represents the design outdoor winter temperature for the heating system of the building. The vertical temperature gradients (P1, P3 and P4 in Figure 4) obtained from this simulated case were compared with simulated temperature profiles presented in earlier work by Risberg et al.\cite{18,38} This case was mainly done to investigate if an acceptable indoor air temperature was possible to achieve with an air heating system and the supply devices installed at the ceiling with disc diffusers. The supply air temperature for this simulation case was $45.4^\circ$C in order to compensate for the transmission and infiltration losses. The value was calculated to achieve an average indoor temperature similar to case 1.

**Result and discussion**

**Annual energy usage**

The annual energy usage (adjusted normal year) for the building is summarized in Figure 6. From the figure, it can be seen that of the amount of solar electricity produced, only around 1/3 can be used in order to reduce the building’s specific energy. Two-third of the produced electricity was sold to the grid, but if people were living in the building, some parts of this could be used as household electricity. The ventilation losses for the building on an annual basis were around 33%. The summed up values for district heating and electricity were used to calculate the specific energy for the building according to equation (1), as presented in Table 6. Therefore, the building fulfills the Swedish demands regarding a passive house for both criteria.

![Figure 6. Energy balance of the building.](image)

**Table 6. Annual energy demand and passive house standard.**

<table>
<thead>
<tr>
<th></th>
<th>Normal year</th>
<th>FEBY</th>
</tr>
</thead>
<tbody>
<tr>
<td>Heat loss factor</td>
<td>18.8 W/m$^2$</td>
<td>19 W/m$^2$</td>
</tr>
<tr>
<td>Specific energy weighted</td>
<td>67.9 kWh/m$^2$</td>
<td>77 kWh/m$^2$</td>
</tr>
</tbody>
</table>


Energy characterization

During the measurement period (28 November 2016 to 15 January 2017), the heat losses are categorized as transmission, infiltration and ventilation losses presented in Figure 8. The ventilation losses are directly connected to the outdoor temperature, since the airflow is constant over time. The temperature efficiency of the heat exchanger was calculated from measured values and is presented in Figure 7. The results show that the temperature efficiency is independent of the outdoor temperature. The constant efficiency was expected, since a rotational heat exchanger was used and would not need any defrosting. Since no people were living in the building during the measurement period, no moisture was added to the building and the energy efficiency is the same as the temperature efficiency with an average value of 79%. 

In order to include the thermal inertia in calculations, an outdoor temperature representing a backward weighted moving average value for the last 10 days was used based on results in Figure 7(b). The figure shows the mean absolute deviation between measured and calculated heat losses (hourly basis) with different weighted outdoor temperatures (0–20 days backward weighting factor). The minimum deviation was at 10 days. To verify the calculated heat demands, the measured heat flows between the 27 November 2016 and 15 January 2017 were used, as presented in Figure 8(a). The standard deviation between measurement and calculation was 68 W and the overall mean value differs by 2 W. The difference between measurement values and calculated values was mainly dependent on the fact that the temperature of the supply air was varying and the calculated values were on an hourly basis. The calculated transmission, infiltration and ventilation losses are presented in Figure 8(b) for the CFD validation period together with the total losses ($Q_{tot}$ calculated). The total air supply to the building was excessive and should be reduced since the maximum air supply temperature 52°C was never reached. Thereby the ventilation losses would also be reduced compared to the measurement results.

The results show that it is possible to predict the heat demand for a building with overall simple equations, if the thermal inertia is taken into account.

![Figure 7](image1.png)

**Figure 7.** (a) Heat exchanger temperature efficiency versus outdoor temperature and (b) deviation between measured and calculated heat losses.

![Figure 8](image2.png)

**Figure 8.** Calculated heating demand for the building compared with the measured one.
through a weighted moving average value of the outdoor temperature. If the backward average moving outdoor temperature cannot be decided from experiment, the calculated building time constant can be used in order to decide the numbers of days.

Validation

A CFD model for the building was constructed and compared with manual measurements of the indoor air temperatures and velocities during one day (14 December 2016). Boundary conditions for the envelope surfaces of the building were created in the simulation from results in the previous section. The predicted temperature gradients from the CFD model correspond well with the measured temperature, see Figure 9(a) and (b). The upstairs floor shows larger deviation near the ceiling in the vertical line P2, especially in the nearest measurement point to the ventilation supply device. The control unit of the supply air temperature did not work correctly. The temperature was all the time fluctuating between 21°C and 45°C in periods of around 9 min. In the occupied zone, the difference between the simulated and measured values was on average 0.2°C. This difference is in the same range as other validation studies, for example the studies by Elshafei et al., which presented how window parameters could have an effect on natural ventilation in a residential building. The study also showed that the temperature difference in horizontal position did not change significantly in a single room.

The velocity at 31 points in the building was measured but in only five of these, the value was within the measurement range of the equipment (above 0.05 m/s). These points V1–V5 are presented in Figure 9(c) where the error bars represent the manufacturer’s specified data for the used instrument. The simulated velocities show a good agreement in three out of the five measuring points. For points 2 and 3, the deviation was outside the measurement tolerance of the hotwire anemometer. The explanation of that is probably that the inlet air temperature was varying a lot, which caused differences in the supply mass flow. The direction of the airflow could also be affected due to buoyancy effects when both colder and warmer airflows pass through the supply devices. The measured values at these points were also far lower than the calibration values for the measurement device. In overall, air velocities below 0.15 m/s would only have a small effect on the thermal comfort according to the Fanger comfort model. Overall, the air velocities were in levels below 0.15 m/s inside the occupied zone, therefore no figures of flow distribution are presented.

Figure 9. Validation results: (a) and (b) air temperatures and (c) air velocities.
Thermal indoor climate simulations

The thermal indoor climate for the different cases (1 and 2) were simulated with the CFD model. The results are presented in a vertical plane at the centre of the left apartment of the building according to Figure 10. Calculations with equations (2) to (5) resulted in heat losses presented in Table 7. During case 1, the energy supplied to the building was registered at 1.5 kW and a supply air temperature of 34.7 °C on average. During case 2, the total heat demand for the building was calculated at 2.5 kW, and the supply air temperature was 45.4 °C in this case.

Simulation case 1. The temperature distribution on the 14 December is presented in Figure 11(a). It can be seen that the vertical temperature gradient in the occupied zone is in the level of 0.4–0.9 °C/m, with the largest difference at the ground floor. However, in horizontal direction, the temperature distribution is practically even. The temperature on the first floor is higher compared to that on the ground floor. The supply airflows were divided; as 63% of the total mass flow on the first floor since more ‘clean’ rooms like bedrooms were situated there. The PPD index presented in Figure 11(b) shows an optimal thermal indoor climate (<15%) on the first floor. Due to the lower temperature on the ground floor, the number of dis-satisfied people increased to between 15% and 24% in certain parts of this volume. For the whole building, the thermal indoor climate in the occupied zone for first floor was satisfied (PPD < 15%). For the ground floor, almost every room has a PPD level above 15%.

If the ventilation flows are rearranged (switch the air amount between the floors) so the main part (2/3) of the total supply airflow is delivered to the ground floor, the temperature distribution (Figure 12(a)) between floors is more evenly compared to the original supply airflow. The large amount of air supply to the living room would create an increased temperature gradient of 0.9 °C/m. In order to fulfill the EN ISO, 7730 standard at category C, the PPD value must be below 15%. This can be achieved with rearranged ventilation flows (Figure 12(b)), and this applies for the majority of the building volume.

In our simulations, the inner doors were open except to the bathroom and laundry. All rooms except rooms

<table>
<thead>
<tr>
<th>Table 7. Data for the different cases.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case</td>
</tr>
<tr>
<td>------</td>
</tr>
<tr>
<td>1</td>
</tr>
<tr>
<td>2</td>
</tr>
</tbody>
</table>

Figure 11. Thermal climate for the Simulation case 1. (a) Indoor temperature. (b) Predicted percentage of dissatisfied (%).
with closed doors show PPD levels below 15% in the occupied zone. Inside rooms with closed doors, it was difficult to fulfill the standard since air was entering the room through a gap at the bottom of the door. Due to the temperature gradient, air of lower temperature entered these rooms. PPD values around 20-30% and PMV around -0.7 occur, which means that the air temperature was too low in these rooms. The results show the importance of more detailed knowledge when designing air-heating systems in buildings. To avoid the problem, transfer devices should be installed above each inner door in order to better distribute the ventilation airflow to each room.

**Simulation case 2.** For the case with an outdoor temperature of -25.7°C (DOWT) and according to ordinary supply airflows, the air temperature distribution in the selected plane is presented in Figure 13(a). The temperature gradients are larger compared to the previously validated case since supply air temperature was increased. The enlarged transmission heat losses from window surfaces make a cold draft on the ground floor. The PPD index is presented in Figure 13(b) where the area with optimal thermal indoor climate was reduced compared to case 1 mainly due to larger temperature gradients. The increased supply air temperature (45.4°C) and original supply ventilation flow rate on the first floor caused the higher temperatures. The lower air temperatures close to the floor at the ground level would decrease the transmission losses towards the soil, since the soil temperature was basically constant during the winter (+4°C), due to a snow layer that isolate against the outside air temperature. An overall increased air temperature at the ground floor compared to the case 1 gives a better thermal indoor climate. Areas close to the entrance door and windows indicate PPD values above 15%. On the other hand, the climate on the first floor was poorer compared to case 1 due to the fact that the air temperature on the first floor became too high. For the whole building, the corresponding behaviour was observed. All rooms with closed doors was still indicating PPD values above 15%.

With adjusted airflows (increased flow on the ground level according to the earlier case) and at DOWT (-25.7°C) the temperature distribution and
PPD values are presented in Figure 14. The figure shows that the first floor has a good thermal indoor climate, but that the increased supply flow on the ground floor results in high temperatures in the living room. For the thermal climate model, these high temperatures have a negative impact. Once again rooms with closed doors display PPD values above the limit for category C and the bedroom and living room on ground floor show high PPD values (>15%), since a large amount of heat supply was in these rooms. Depending on only supply airflows to two different rooms (two supply devices in the shown living room) on the ground level, high air temperatures were generated in these rooms. A more even distribution of the supply air devices on ground floor should properly reduce these negative effects. In order to fulfil a good indoor climate with PPD values below 15%, the amount of supply airflow and location of the devices must be carefully considered.

**Comparison with different heating systems.** Earlier work with simulations of a low energy building has been presented by Risberg et al.\textsuperscript{18,38} Temperature gradients from this work, the radiator system and an underfloor heating system are presented in Figure 15. The plotted gradients were located in a living room (with a supply air device) 1 m in front of a window. The vertical air temperature distribution for an air heating system (from current work) is also included in the figure. The outdoor air temperature in all simulated cases was –30°C and the building types are comparable. In the actual building, a room with exhaust airflow (kitchen) is represented through line P4 in Figure 4, rooms with supply air are represented through line P3 (living room on the ground floor) and line P1 (family room on the first floor, this deviates from other measurement points by the occurrence of a slightly positive heat flux through the floor). Figure 15 shows the temperature gradient in rooms with exhaust

![Figure 14](image1.png)

**Figure 14.** Thermal climate for the Simulation case 2 with rearranged ventilation flows. (a) Indoor temperature. (b) Predicted percentage of dissatisfied (%).

![Figure 15](image2.png)

**Figure 15.** Temperature gradients for different heating systems at an outdoor temperature of –30°C.
air was smaller than in cases with supply air. Room with an exhaust airflow also have smaller deviations compare to radiator heating system, which is the most common used heating system in Sweden. The floor temperature for air heating system was higher than the air temperature close to the floor, which is due to heat radiation to the floor from other surfaces, mainly the ceiling in the room. Figure 15 also illustrates that the temperature gradients in rooms with supply airflow were somewhat higher, compared to heating systems using radiators. Overall, a similar temperature gradient can be achieved with an air heating system and a radiator heating system.

**Thermal climate evaluation.** The building was categorized according to EN ISO standard 7730, Category C, with results for the main part of the building presented in Table 8. The three rooms with closed doors were excluded, since the air temperature in these rooms was always too low. With the original airflow distribution, Category C was fulfilled in case 1 on the first floor and the opposite in case 2. The changed results are due to differences in heat demand. When the airflow distribution was rearranged, category C was fulfilled for both levels in case 1. In case 2, there was still some malfunction on the ground floor.

In total, in all investigated cases, the PMV value was between -1 and +1, which means that the thermal indoor climate was only slightly too hot or cold in the whole building volume. The other criterion for DR and PD was fulfilled with category A in both cases except in the top part of door openings where the main part of the warm airflow passes between rooms. The values according to PPD and PMV are hard to fulfill in the whole building volume, mainly because of temperature differences between these two floor levels. More even heat distribution could occur in buildings with a radiator heating system that has heat supply to each room, since thermostat valves can handle increased temperatures. The radiator system also has a higher inertia to transfer heat to the air in the building. An underfloor heating system where no temperature gradient appears would give a more even heat distribution in the room.

An estimate of the thermal indoor climate in a building from only one point in the indoor volume can be precarious. From the simulations done in the paper, it is possible to find spots where PPD values below 6% can be detected. In order to describe if a building has a good indoor thermal comfort, it is important to study the total indoor volume. This applies especially when local distributed systems like air heating systems are used and for buildings with more than one floor level. According to the Swedish building requirements, the thermal indoor climate of the building fulfills the demands.

**Conclusions**

It is possible to build a passive house (according to Swedish demands) even at a geographical location in a sub-Arctic region. The analysis conducted for the building, found a heat loss factor of 18.8 W/m² floor area and an annual specific energy usage of 67.9 kWh/m² floor area.

The heat demand of the building can be characterized with basic equations, if the thermal inertia of the building is included. It can be done through a backward weighted moving average value of the outdoor temperature but requires measurements of the energy supply to the building for a short period of time.

The temperature distribution in a building can be predicted using CFD simulations with the k-ε model. Validation with manual measurements shows good agreement with simulations. The air temperature and airflows between different rooms and floor levels were described in a correct way. The technique can be used to present the thermal indoor climate from a human perspective for the whole building volume.

For the EN ISO, 7730 standard the PPD and PMV are hard to fulfill when a locally distributed air heating system is used. Large temperature differences between floor levels could be caused by excessive supply airflow to the first floor.

The results show that an air heating system can create an acceptable thermal indoor climate. Simulations show that it is possible to improve the thermal climate by rearranging the amount of supply air. The temperature gradient between floor and ceiling

<table>
<thead>
<tr>
<th>Original airflow distribution</th>
<th>Case 1</th>
<th>Case 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ground floor</td>
<td>×</td>
<td>✓</td>
</tr>
<tr>
<td>First floor</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Rearranged airflow distribution</td>
<td>Case 1</td>
<td>Case 2</td>
</tr>
<tr>
<td>Ground floor</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>First floor</td>
<td>✓</td>
<td>✓</td>
</tr>
</tbody>
</table>

PPD: predicted percentage of dissatisfied; PMV: predicted mean vote; DR: draft ratio; PD: local thermal discomfort.
in a room is comparable to gradients established with a radiator heating system. According to Swedish building regulations, an acceptable thermal indoor climate during the winter period is achievable in a passive house located above the Arctic Circle. Using an air heating system with supply devices located on the ceiling has been shown to work well also at the design outdoor winter temperature.

The simulations show that in order to describe the thermal indoor climate, one needs to include the total volume, not only one point, which normally is obtained with building simulation softwares.

Authors’ contribution
D. Risberg planned and performed the simulations and calculations with support by the co-authors. All authors jointly analysed the results and summarized the conclusions in the article. All authors read, modified, commented and approved the manuscript.

Acknowledgements
Cooperation has taken place with NCC, ABELKO and our academic partners at Luleå University of Technology, who made this research possible.

Declaration of conflicting interests
The author(s) declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

Funding
The author(s) disclosed receipt of the following financial support or the research, authorship, and/or publication of this article: This work was mainly funded by ATTRACT (Attractive, sustainable living in cold climates), a research program financed by VINNOVA (the Swedish Innovation Agency). The author(s) disclosed receipt of the following financial support or the research, authorship, and/or publication of this article: All authors read, modified, commented and approved the manuscript.

References
Paper V
The impact of snow and soil freezing for commonly used foundation types in a subarctic climate.

Daniel Risberg1*, Mikeal Risberg1, Lars Westerlund1

1 Division of Energy Science, Lulea University of Technology, Lulea, Sweden

*Corresponding author: Daniel Risberg, Luleå University of Technology, SE-971 87 LULEÅ, Sweden. Email: daniel.risberg@ltu.se

Abstract

Heat losses from a building foundation are affected by both the surrounding conditions and the surrounding soil properties. These include many factors that complicate the analysis of heat loss, such as thermal storage, snow and soil freezing. The effect of snow and soil freezing was studied with a 3D simulation model in a subarctic climate.

The heat losses from the most commonly used foundation types in Sweden were studied. This paper shows that it is possible to achieve a good thermal estimation of the air temperatures in a crawl space, with an average difference of 0.4°C compared with the validation data over a year. Snow and soil freezing reduce the annual heat losses through the different foundation types by 7-10% and the maximum heat loss rate by 13-25%. In order to describe the heat transfer correctly, snow must be included in the calculations, while soil freezing has only a minor impact. The 3D model implemented in this study shows a significant impact on the soil temperatures when these parameters are included.

For a subarctic climate, the commonly used calculation methods based on the European standard ISO 13370 are not thorough enough to calculate the heat transfer through a foundation accurately.

1 Introduction

Heat losses from a building through the foundation are a complex phenomenon which includes many different parameters. The losses are affected by both the environmental conditions and the surrounding soil properties, which also include the multidimensional nature of most earth-coupled heat transfer processes [1]. Overall, heat losses through the foundation of a building are an area which has received comparatively little attention. Heat losses from building envelopes like walls and roofs are a well-known phenomenon which is often simulated with good agreement using one-dimensional (1D) heat transfer. This approach
is used in most software for the simulation of heat losses from buildings [2] and is validated against EN ISO 13791 [3] with good agreement. For heat losses through the foundation, some standard cases are described in the standard ISO 13370 [4], which provides a simplified approach that does not consider in detail all the parameters affecting the heat transport. A three-dimensional (3D) heat transfer simulation is more suitable and accurate compared with a 1D simulation for heat losses through the foundation of a building, since more heat transfer mechanisms can be included.

In total, Sweden has around two million single-family houses [5]. Three types of foundation construction dominate the market, namely crawl space, slab and basement foundations, each of which has around 25-35% of the market [6]. The basement type was used especially several decades ago. In newly built houses the slab foundation is the most common option, while crawl space foundations are mainly used for modular housing when the buildings have already been constructed with a joist floor. The main reasons why the crawl space foundation is a common type for buildings are that it provides ease of construction in hilly terrain and offers good possibilities of ventilating radon from the ground. The construction exhibits problems connected with a high moisture content during the summer period, when the warm outdoor air is cooled by the lower soil temperature under the foundation. The high moisture content (above 70% RH) in crawl spaces could result in the problem of mould growth. The slab foundation construction is used mainly because it is associated with fewer moisture problems and because it facilitates the installation of a water-based underfloor heating system, which makes it a popular type of foundation [7].

Some parts of Sweden are located in subarctic regions where the snow layer and soil freezing will affect the heat losses from the foundation. With air temperatures below zero for a large part of the year, frost heave can arise and temperatures below zero beneath the foundation construction imply an essential risk of frost heave, mainly caused by ice lenses. The problem of frost heave in Sweden occurs especially in soils of silt or moraine. One important method of preventing frost heave is to dig down to a frost-free depth and add filling material. Many newly built houses have frost protection insulation outside the foundation in order to reduce the filling material.

Nowadays new buildings are designed with intending to decreasing the energy usage and extra-insulated foundations will be more common in the future. The increased popularity of energy classifications of buildings according to the passive house standard is making it more important to consider and predict the heat losses from the foundation more accurately.

Different studies of how to treat earth-contact heat transfer were reviewed in a paper by Zoras et al. [8], who concluded that 3D simulation of a house foundation is the most accurate method for describing the heat losses correctly. A complete tool must therefore deal with variable thermal conductivity, and include heat and moisture coupling, phase changes, snow cover and convection and evaporation at the soil surface outside the building.
Weitzmann *et al.* [9] created a dynamic 2D model for the simulation of a slab foundation (with and without underfloor heating), and they scaled the results with a characteristic dimension, showing good agreements with measurements. Mitalas *et al.* [10] showed that to include the corner effects for the foundation, it is important to use a 3D model. Janssen *et al.* [11] studied the influence of soil moisture, and their results show that it is important to include the water in the soil to predict correctly the heat conductivity of the soil in a simulation model. The influence of soil freezing was studied by Bligh and Willard [12], who created a model in which snow was excluded according to the geographical location. A paper by Rees *et al.* [13] also investigated soil freezing affected by earth-contact heat transfer, and the temperatures outside a foundation were considered. In the model created by Rees *et al.* [13], snow was included, the heat conductivity, density and heat capacity were constant, and the snow depth was never greater than 0.2 m. The conclusions of the paper were that it is important to capture the soil surface boundary conditions, of which snow is a significant factor.

The present paper presents an improved method for treating snow and include heat resistance. This is accomplished by simulating the heat losses with a 3D conductive numerical model. At the studied location, the snow period is of a substantial length and the snow depth was above 1 m during part of the year. The paper contributes to an enhanced understanding of how snow and ground freezing affect the foundation heat losses. Moreover, common foundation types in Sweden are studied and compared. The 3D model developed in this study was also compared with a model based on the standard ISO 13370, a standard used for many software packages for building energy simulation. The model created was validated with measurements of air temperatures inside a crawl space foundation located in Luleå in the north of Sweden.

The aim of the study was to investigate how the thermal losses from a common building foundations are affected by the cold climate typical of a subarctic region.

2 Theory

In a cold climate the soil temperature is affected by the thermal insulation from the snow layer and soil freezing. The calculations of the frost penetration depth in the simulation model are based on the thermal conductivity and heat capacity, which are dependent on the temperature in the soil. The frozen and unfrozen thermal conductivity of the soil was calculated according to Johansen’s method [14]. The dependence of the heat capacity and thermal conductivity on the soil temperature is presented in Figure 1. The soil properties have the same conditions as those reported in Pericault *et al.* [15], i.e. a soil water content of 0.09 m$^3$/m$^3$, a porosity of 0.4 m$^3$/m$^3$ and a dry density of 1600 kg/m$^3$. 

3
The snow’s thermal insulation is dependent on the snow density, which is difficult to measure continuously. The snow layer density on the ground is often the sum of many different densities. New snow has a density dependence on both the temperature and the wind speed.

The density of the snow is calculated from the water equivalent and average values for the snow depth. An assumption is made that no water is removed from the snow before the snow density has increased above 550 kg/m³, snow density and water equivalent is presented in Figure 2. The thermal conductivity (Equation 1) in the model is based on a polynomial fit to measurements performed by Kellett et al. [16].

$$\lambda_{\text{snow}} = 4 \cdot 10^{-6} \cdot \rho_{\text{snow}}^2 - 0.0013 \cdot \rho_{\text{snow}} + 0.2122$$

Figure 1. Soil properties versus temperature.

Figure 2. Equivalent water mass and snow density during the snow period.
The density affects the thermal inertia of the snow layer, together with the specific heat capacity of the snow. The specific heat capacity of the snow was specified as 2.08 kJ/kg K. During the main part of the winter any heat transfer to the snow is not causing melting, melt water is refreezing in the snow layer until the snow is fully saturated and pores in the snow is filled which increase the density of the snow [17]. Water as liquid is started to be removed by melting when the snow layer gets a density of around 550 kg/m$^2$, the snow layer consists of slush formed of snow and water. The snow temperature is at 0℃ during the melting. In order to keep the snow temperature at 0℃ during outdoor air temperatures above zero degree a peak in the specific heat capacity was added for the temperature range -0.15 to 0℃ (in order to include latent heat). These properties are implemented in the 3D model.

2.1 ISO 13370 Standard

The $B'$ value in Equation 2 represents the characteristic dimension of the floor in the building, while $A$ represents the floor area inside the outer walls and $P$ is the perimeter of the floor area.

$$B' = \frac{A}{0.5P}$$  (2)

The average monthly heat flow rate from the building foundation to the environment, $\Phi_m$ (W), is given by Equation 3 [4]. Here the parameter $H_g$ (W/K) is the steady-state overall heat transfer coefficient, $H_{pi}(W/K)$ is the internal periodic overall heat transfer coefficient, and $H_{pe}(W/K)$ is the external periodic overall heat transfer coefficient. Changes in the indoor conditions in the building are treated by the internal parameter and changes in the outdoor conditions are treated by the external parameter, for the purpose of including the thermal inertia in the calculations. The $\alpha$ parameter is the time lead in months of the heat flow cycle compared with that of the internal temperature, and $\beta$ is the time lag in months of the heat flow cycle compared with that of the external air temperature.

The $\bar{\theta}_e$ and $\bar{\theta}_i$ parameters describe the amplitude of the temperature variation for the external and internal temperatures. With a constant indoor temperature, $\bar{\theta}_i$ becomes zero and the second part in Equation 3 can then be excluded. In Equation 3 the parameter $m$ stands for the number of the month and $\tau$ stands for the number of the coldest month of the year.

$$\Phi_m = H_g(\bar{\theta}_i - \bar{\theta}_e) - H_{pi}\bar{\theta}_i \cos(2\pi \frac{(m-\tau-\alpha)}{12}) + H_{pe}\bar{\theta}_e \cos(2\pi \frac{(m-\tau-\beta)}{12})$$  (3)

The 1D simulation programs use a method based on a soil depth of 0.5 m, with a virtual layer of 0.1 m outside of the soil. This computational domain is set outside the foundation of the building. For the virtual area, a thermal conductivity is set without any density and
heat capacity values. The virtual layer is used to compensate for additional heat resistances. This method is used, for example, by the simulation program IDA ICE.

The $H_g$ value is calculated for the 1D simulation programs according to Equation 4. For the bottom part of the slab foundation, the overall heat transfer is calculated using Equation 5, and for the foundation walls, the $U_{bw}$ value is calculated according to Equation 6. The $H_{pe}$ value is calculated according to Equation 7.

\[
H_g = U_{bf}A + zP U_{bw} \tag{4}
\]

\[
U_{bf} = \frac{2\lambda}{\pi\delta + d_t + 0.5z} \ln \left( \frac{\pi\delta'}{d_t + 0.5z} + 1 \right) \tag{5}
\]

\[
U_{bw} = \frac{2\lambda}{\pi z} \left( 1 + \frac{0.5d_t}{d_t + z} \right) \ln \left( \frac{d_t}{d_w} + 1 \right) \tag{6}
\]

\[
H_{pe} = 0.37 \cdot PL e^{-\frac{z}{\delta}} \ln \left( \frac{\delta}{d_t} + 1 \right) + 2 \left( 1 - e^{-\frac{z}{\delta}} \right) \ln \left( \frac{\delta}{d_w} + 1 \right) \tag{7}
\]

The $z$ value represents the depth of the ground (if $z$ equals zero, it represents a slab foundation). The parameter $d_t$ represents the total equivalent thickness downwards, and $d_w$ denotes the total equivalent thickness for the foundation walls. The parameter $\lambda$ is the thermal conductivity of the soil and $\delta$ is the thermal penetration depth.

In IDA ICE the temperature at the outside of the virtual layer ($T_v$) is calculated according to Equation 8.

\[
T_v = T_0 + c T_{Amb} \tag{8}
\]

In Equation 8, $c$ is calculated according to Equation 9, $T_0$ according to Equation 10 and $T_{Amb}$ is the monthly mean ambient outdoor temperature with a phase shift based on $\beta$.

\[
c = \frac{H_{pe}}{H_g}, \text{ if } c < 1, \text{ otherwise } 1 \tag{9}
\]

\[
T_0 = T_{mean} (1 - c) \tag{10}
\]

In Equation 11, $T_{mean}$ is the annual average outdoor temperature.

3 Method

A 3D simulation is presented in the first section (3.1), the 1D model in section 3.2 and the climate data (section 3.3) was used for both models. Measurements for validation of the 3D model is described in section (3.4) and in the last section data for a passive house is declared. The 3D transient model was used to decide the heat losses from the different foundation types. The model was validated against measured air temperatures in a crawl
space located in the city of Luleå, in the north of Sweden. The measurement period was one year and lasted from 8th May 2014 to 8th May 2015.

3.1 Thermal model

The model was built as a 3D transient simulation model in Ansys CFX 18.0. The fluid flow for the air in the crawl space was assumed in the model to be well mixed (with the same temperature in the fluid domain for each time step) and no flow equations were solved in the model; this approach was adopted to be able to include radiation between the surfaces. The time step in the calculations was specified as one day. The outdoor air temperature and snow depth boundary conditions were set according to weather data from SMHI (the Swedish Meteorological and Hydrological Institute) [18]. The model was first run for a whole year to achieve an approximate distribution of temperatures in the soil due to its high thermal inertia. These soil temperatures were then used as initial values for the actual simulation. The conservation of energy equations was solved numerically with a convergence criterion for the residuals of less than 1e-6 for the root mean square value in the simulation volume.

The radiation between the vertical walls inside the crawl space was determined by the discrete transfer method [19]. Since slab foundations do not include any air volume, a radiation model was excluded during simulations of this foundation type.

3.1.1 Geometry

The foundation of the building had an inner dimension of L*W: 11.26x7.06 m. The simulation volume had a horizontal width of 5 m, measured from the outside of the foundation, and a depth of 10 m. A schematic figure of the simulation setup is presented in Figure 3. The $B'$ value of the foundation was calculated according to Equation 2 to be 4.34 m.

3.1.2 Boundary conditions

An adiabatic boundary was set at the bottom of the simulation model. Symmetry was used for the vertical sides of the simulation domain, since the building was located in a residential area with an approximate distance to the neighbouring buildings of 10 m. For the crawl spaces, a system of joists was included as a thermal resistance with a value of 4.35 K m²/W. The floor temperature inside the building was for all the cases set to a constant value of +20°C. The heat transfer coefficient between the outdoor air and the parts of the foundation walls in contact with each other was set to 7 W/m² K (including both convection and radiation), and this value was also used for the snow layer or the soil in the area of contact with the outdoor air. For the inner surfaces of the crawl space, the emissivity was set to 0.9, and the convective transfer coefficient on the walls was set to 2.5 W/m² K and 0.7 W/m² K for the ceiling and the bottom surface, respectively. The heat conductivity for all the materials was set according to Table 1.
Figure 3. Geometry of the simulation volume.

In total, four different foundation types were compared. The different types examined were an uninsulated crawl space foundation, an insulated crawl space foundation, an insulated slab foundation and a slab foundation with extra insulation against the vertical edge of the foundation in order to break thermal bridges. The cross-sections of the different foundation types are presented in Figure 4. The numbers in the figure represent different materials, whose data are presented in Table 1, and the dimensions for the different foundation parts are presented in Table 2. All the compared foundation types were simulated using the same $B^*$ value. The heat losses from each foundation were determined as the heat flux through the floor of the building.
Figure 4. Different foundation configurations: a) uninsulated crawl space, b) insulated crawl space, c) insulated slab, d) slab with extra insulation.

Table 1. Material data.

<table>
<thead>
<tr>
<th>Number</th>
<th>Material data</th>
<th>Thermal conductivity (W/mK)</th>
<th>Density (kg/m³)</th>
<th>Heat capacity (J/kgK)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Soil</td>
<td>See Figure 1</td>
<td>1600</td>
<td>See Figure 1</td>
</tr>
<tr>
<td>2</td>
<td>Concrete</td>
<td>1.9</td>
<td>2300</td>
<td>900</td>
</tr>
<tr>
<td>3</td>
<td>Insulation</td>
<td>0.033</td>
<td>35</td>
<td>750</td>
</tr>
<tr>
<td>4</td>
<td>Air</td>
<td>1000</td>
<td>1.2</td>
<td>1004</td>
</tr>
<tr>
<td>5</td>
<td>Snow</td>
<td>Equation 1</td>
<td>See Figure 2</td>
<td>2080</td>
</tr>
</tbody>
</table>
Case A was used for validation of the simulation model. During the measurements for the validation, a dehumidifier was used in the foundation, and in order to compensate for this in the simulations, a heat source was placed in the simulation volume with a power based on measurement values. This case was also simulated without the dehumidifier, and the amount of heat which the building gained from the dehumidifier could thereby be studied. In this paper, all the data for the crawl space are presented without a heat source installed, except for the data for the validation case. The model without heat from the dehumidifier was compared with a frequently used software package for building simulations, IDA ICE, which uses the conventional ISO 13370 standard, in order to compare a 1D model with the presented 3D model.

The annual heat losses for the different foundation types were simulated and compared. It was investigated how a snow layer and soil freezing (included in a total model) affect the heat losses. The different foundations were also studied in order to compare how snow and ground freezing influence the classification for a passive house.

We investigated what would happen if frost protection insulation outside the foundation was not included for case C and D, in order to examine if the soil temperature would fall below zero degrees (entailing a frost heave risk) under the foundation.

### 3.2 IDA ICE model

The IDA ICE model that was used for comparison is a 1D model which is based on the ISO 13370 standard and where cold bridges are treated as a heat loss factor (W/K) based on the outdoor temperature. The standard does not consider snow and soil freezing. In order to compare the methods, a 3D model without snow and soil freezing was used. Foundation type C and D were not compared, since the IDA ICE software does not offer the possibility of considering vertical insulation for slab-on-ground foundations.

The software calculated the $H_s$ value according to Equation 5 and, on that basis, determined the virtual layer value. The first term in Equation 5 was used for the crawl space soil boundary condition facing downwards, while the second term was used for the vertical parts of the crawl space.

### Table 2. Dimensions of foundation parts according to Figure 4.

| L1=17.5cm | H1=25.0cm | H6=10.0cm |
| L2=5.0cm  | H2=15.0cm | H7=10.0cm |
| L3=10.0cm | H3=75.0cm | H8=10.0cm |
| L4=20.0cm | H4=10.0cm | H9=30.0cm |
|           | H5=10.0cm |            |
The soil and foundation properties from foundation type A gave an $H_g$ value of 69.224 W/K and an $H_{pe}$ value of 38.589 W/K from the IDA ICE software. The virtual layer conductivity was set to 1.39e7 W/m K for the walls and 0.0708 W/m K for the bottom.

For foundation type B, an $H_g$ value of 12.52 W/K and an $H_{pe}$ value of 6.12 W/K were given by the software. The virtual layer conductivity was 1.281 W/m K for the walls and 0.0557 W/m K for the bottom.

The ISO 14683 standard gives default values for thermal bridges that overestimate the heat losses (with errors in the range of 0 to 50%) and, furthermore, not all the foundation types are described in the standard [20]. The bridges in the simulations were set to give the best fit to the 3D simulation model. This was done in order to be able to compare the models without the influence of the values chosen for the thermal bridges.

### 3.3 Climate data.

The climate data used in both models were the outdoor air temperature and snow depth (daily average values) was only used in the 3D-model, derived from meteorological data issued by SMHI [18]. From the snow depth, the thermal conductivity was calculated according to Equation 2. The annual variations are presented in Figure 5. It can be seen that the daily mean outdoor temperature varies between +25.6°C during the summer and -23.6°C during the winter. The snow depth was above 1 m during a short period in the chosen year, and for 168 days of the year there was a snow layer on the ground.

![Figure 5. Environmental conditions – outdoor air temperature (°C) and snow depth (m).](image)

### 3.4 Measurements.

The measurements were performed with eight calibrated temperature gauges inside the crawl space, and the sampling time used was 15 minutes (average value). The sensors were
located at a distance of 10 cm beneath the joist and placed in the crawl space according to Figure 6. The total vertical depth of the crawl space was 0.75 m. The average air temperature value in the crawl space was calculated based on the gauge data, and the 15-minute values were also recalculated to average daily temperatures over the whole year. The electricity supplied to the dehumidifier was measured manually at certain times, and the average value on a daily basis was calculated.

Figure 6. Measurement setup (temperature gauges’ location).

3.5 Passive houses

The results from the simulations were used to investigate how snow and ground freezing influence the classification of passive houses. The passive house standard requires the fulfilment of two criteria, a specific energy usage of less than 63 kWh/m² of floor area and a maximum heat loss factor of 19 W/m² at the design outdoor temperature [21]. Common data for a passive house in Sweden were used and both one- and two-storey houses were compared.

The ventilation heat exchanger temperature efficiency was set to 85% with a ventilation flow rate according to the Swedish requirements of 0.35 l/s m² of floor area. The building infiltration rate was set to 0.3 ACH with a pressure difference of 50 Pa, which represents a typical passive house value. The real case infiltration rate is approximately 5% of this value [22], which gives an infiltration rate of 0.015 ACH. The building envelope represented that of a typical passive house, with a U value of 0.10 W/m² K for the walls, 0.080 W/m² K for the ceiling, 1.0 W/m² K for the doors and 0.80 W/m² K for the windows. The envelope had an area for the floor and ceiling of 79.45 m². The outer wall area was 86 m² for the one-storey house and 180 m² for the two-storey house. The window area was approximately 15% of the floor area (12 m² per floor) and the entrance door had an area of 2 m². The specific energy usage was calculated with standard values for the hot water and household electricity [23]. The effect of the occupants and solar radiation was set according to the degree-day method.
4 Results and discussion

4.1 Validation and verification

The total model, including both snow and soil freezing, was validated against measurements of the air temperature inside the crawl space. When the measurements were being performed for the validation, with a dehumidifier running in the crawl space, the thermal losses through the floor of the building decreased due to the heat (from the dehumidifier) added to the air in the crawl space. For a correct comparison of the different foundation types used in this study, no extra heat can be added to the crawl space. For validation of the total simulation model, the heat from the dehumidifier was introduced as a heat source.

In Figure 7 the black line represents the simulated air temperature and the blue dashed line represents the measured daily average air temperature in the space over a year. The studied period lasted from 8th May 2014 to 8th May 2015 (measurement data were missing for the period from 1st to 24th January). The average outdoor temperature for the studied period was 4.4°C, while the normalized outdoor temperature for a 30-year period (1961 to 1990) is 1.6°C. The quantity of supplied heat was set in the model according to the measured amount of electricity supplied to the dehumidifier. The results show a good agreement between the simulation and the measurements of the air temperature in the crawl space over the year. The maximum difference was 1.4°C and the average difference was 0.4°C. This level of agreement indicates that the snow closest to the house cannot have melted during the winter; such melting can be caused by heat transfer from the building.

The model (working on a daily average basis) was also verified by changing the time step size for the climate data in the simulations to an hourly basis. Over the year the mean deviation between the results for the different time bases was only 0.2°C, with a maximum value for a few hours of 1.0°C. Therefore, a daily based configuration is adequate for resolving the temperature change and the heat transfer to the environment over the year. Compared to an hour-based evaluation, the CPU time decreased approximately 24 times for the geometry studied when daily time steps were used.

The grid was refined by increasing the number of elements from 1.01 million to 4.69 million in order to estimate the discretization errors. The difference in the results was very small, with an average deviation of 0.02°C and a maximum deviation of 0.4°C between the different meshes.
Figure 7. Simulated temperatures (black line) and measured temperatures (blue dashed line) in the crawl space.

The simulation model included some uncertainties. For example, the power to the dehumidifier was given as an average value for the different manual sampling times, and the meteorological data were collected at a distance of around 10 km from the building in question.

4.2 Effect of a dehumidifier on thermal losses.

Figure 8a shows the simulated air temperature in the crawl space (foundation type A) with a heat source installed (black line) and without one (blue dashed line). A heat source gives an increased temperature in the crawl space, and the quantity of supplied heat was set to a value according to the measured amount of electricity supplied to the dehumidifier. Figure 8b show the reduced total rate of heat loss for the building through the floor (red dashed line) when the dehumidifier was running. The figure also shows the electricity supplied to the dehumidifier. It can be seen that the main part of the supplied energy does not contribute to decreasing the heat losses of the building. Over the year, around 16% of the supplied electricity reduces the heat losses from the building (89 kWh of a total of 562 kWh), while the remaining part generates heat lost to the surroundings. The supplied electricity causes an increased soil temperature around the foundation. During the last part of the investigated year, the dehumidifier was switched off (i.e. no electricity was supplied from 30th January to 23rd May), but there was still some reduction in the heat losses caused by the thermal inertia of the soil.

In the investigated case, a condensing dehumidifier was used, but other types of dehumidifier are also common, such as desiccant dehumidifiers and thermal dehumidifiers, which
raise the temperature in the crawl space to avoid a relative humidity above 70%. The simulation model could be used to study these types of dehumidifier as well.

**Figure 8.** Comparision between the results obtained with a dehumidifier and those obtained without one: a) temperature difference, b) decreased heat losses and mean electricity supply to the dehumidifier.

### 4.3 Influence of the snow layer and the freezing of water in the soil.

Figure 9a shows the air temperatures in a crawl space (foundation type A) from simulations with snow and soil freezing (black line). Excluding the snow layer causes a decreased temperature in the crawl space (blue dashed line) during the winter. Ignoring both the snow layer and soil freezing in the simulations decreases the air temperature even further (grey dashed line). The grey dashed line in Figure 9a is almost always covered by the blue dashed line, which implies that the influence from soil freezing is small. The snow depth is represented by the orange line. Omitting the snow layer increases the annual heat losses by 129 kWh (7% of the total heat losses through the foundation), while neglecting the soil freezing too only causes an additional increase in the annual heat loss of 21 kWh (1% of the total heat losses). The air temperature in the crawl space using these different approaches is at its maximum around 4 degrees, which affects the design of the heat supply system, since the indoor air temperature in the building is constantly 20-21°C. Snow on the ground is an important aspect to include, since it reduces the peak load of the heating system in the building by 66W, which results in a change in the rate of heat loss through the foundation of 21%.
Figure 9. Influence of the snow layer and soil freezing: a) temperature difference for the crawl space (foundation type A); b-e) heat flux with and without a snowlayer for the different foundations.
The difference in heat flux during the year for the total model was compared with that for the case without snow and ground freezing. This comparison was performed for all the investigated foundation types and the results are presented in Figure 9 b-e. The largest impact on the heat flux occurs during the period from the end of December to the end of February for foundation types A to C, since the impact from the snow layer affects the heat transfer rate. For type D the difference decreases due to increased insulation and the maximum rate is more than halved compared to that for type A.

The annual heat losses for the altered conditions were compared for the studied foundation types. The results are presented in Table 3. The percentage difference during the snow period (15th November to 25th April) is also included. From the results it can be seen that the snow and soil freezing reduced the heat losses for the different foundation types by 7.1-10.0%. For the period with snow the difference was 10.5-16.8%. Foundation type B exhibits the largest difference, i.e. 10.0% over the year (16.8% during the snow season), which is a result of the increased thermal bridge that appears around the lower part of the concrete beam, which was not insulated. Foundation type D has extra insulation to avoid thermal bridges, is overall most insulated and, therefore, shows the smallest difference between the results obtained with and without snow and soil freezing.

The differences between the maximum heat flow rates were also studied and the results are presented in Table 4. These results show the same behaviour as those for the annual heat losses, with a maximum difference of 25% for foundation type B and a minimum difference of 12% for type D. In absolute values, foundation type A exhibits the largest difference of 66W.

Table 3. Annual heat losses with and without the snow layer included.

<table>
<thead>
<tr>
<th>Case</th>
<th>Annual heat losses [total model] (kWh)</th>
<th>Annual heat losses [without snow and soil freezing] (kWh)</th>
<th>Difference (%)</th>
<th>Difference (%) During winter</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>1852</td>
<td>2002</td>
<td>7.5</td>
<td>12.2</td>
</tr>
<tr>
<td>B</td>
<td>1133</td>
<td>1259</td>
<td>10.0</td>
<td>16.8</td>
</tr>
<tr>
<td>C</td>
<td>1432</td>
<td>1559</td>
<td>8.1</td>
<td>13.1</td>
</tr>
<tr>
<td>D</td>
<td>1033</td>
<td>1112</td>
<td>7.1</td>
<td>10.5</td>
</tr>
</tbody>
</table>
### Table 4. Maximum heat flow rates with and without the snow layer included.

<table>
<thead>
<tr>
<th>Case</th>
<th>Maximum heat flow rate (W)</th>
<th>Maximum heat flow rate [without snow and soil freezing] (W)</th>
<th>Difference (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>308</td>
<td>374</td>
<td>17.6</td>
</tr>
<tr>
<td>B</td>
<td>171</td>
<td>228</td>
<td>25.0</td>
</tr>
<tr>
<td>C</td>
<td>227</td>
<td>268</td>
<td>15.3</td>
</tr>
<tr>
<td>D</td>
<td>138</td>
<td>159</td>
<td>13.2</td>
</tr>
</tbody>
</table>

### 4.4 Comparison with a model created using the ISO 13370 standard

The simulations of the crawl space foundations (type A and B) using the 3D model were compared with corresponding simulations performed using a 1D model based on Annex E of ISO 13370, “Application to dynamic simulation programs”. The building simulation software IDA ICE was used for the comparison and the results are presented in Figure 10. The soil and foundation properties were the same for the 3D and 1D models. The thermal bridges (mainly in the corners and at the connection between the edge beam and the soil underneath) were in IDA ICE for foundation type A set to 6 W/K in order to give the best fit compared to the results of the 3D simulation model without snow and soil freezing. The thermal bridges are difficult to predict for the 1D model without numerical simulation. A typical value in the IDA ICE software is 5.37 W/K, and according to the ISO 14683 standard, the value is 7.63 W/K (a difference of 27%).

Foundation type B (the insulated crawl space foundation) was simulated based on the same principle (Figure 10b). The thermal bridges were in IDA ICE set to 4.05 W/K in order to give the best fit. This construction type is not dealt with in the ISO 14683 standard and the typical values from the software are the same as those for foundation type A.
Figure 10. Simulation using the 3D model compared with simulation using the IDA ICE model – air temperature in the crawl space: a) foundation type A, b) foundation type B.

When comparing the 3D simulation without snow and soil freezing with the simulation obtained with the IDA ICE model (a 1D model), one can observe that the deviation is small. The largest differences are in the period of late winter and early spring; the soil has been cooled during the winter, which cannot be resolved correctly with the 1D model.

The best agreement between the total model and the IDA ICE model occurs during the period between 8th May and 17th December, when there are no effects of snow. During the winter, when the outdoor temperature is decreasing and the soil is frozen and insulated by snow, the difference is largest because the ISO 13370 standard does not take snow and soil freezing into account.

The annual heat losses for the IDA ICE cases were 2098 kWh for foundation type A (2002 kWh for the total 3D model) and 1319 kWh for type B (1259 kWh for the total 3D model), which represents a difference between the models of 4.7% for both cases. Overall, a one-dimensional model according to the ISO 13370 standard can predict the heat losses for a crawl space in more southern regions where snow and soil freezing do not appear. For acceptable results, correct values for thermal bridges must be specified. For regions with snow and temperatures below zero degrees during the winter, the difference between the 1D simulation (using IDA ICE) and the 3D simulation of the total model was 13% for foundation type A and 19% for foundation type B, with “properly determined” values for cold bridges used for the 1D simulation. The results show that snow and soil freezing must
be considered for a correct determination of heat losses, which is not possible using the ISO 13370 standard.

4.5 Thermal losses

After validation of the total model, the heat losses for the different foundation types were studied for a whole year. Figure 11 presents the thermal flux based on the daily average outdoor temperature. From the figure it can be seen that the uninsulated crawl space foundation (A) has the highest heat losses, followed by the conventional slab foundation (C). The best performance was exhibited by the extra-insulated slab foundation (D), followed by the insulated crawl space foundation (B). The results are summarised in Table 3, where it can be observed that the annual heat losses for type A are almost double those obtained for type D. The crawl space foundation with insulation (B) reduces the heat losses to almost the same level as that of the losses for foundation type D.

Increased insulation at the vertical edge of the slab foundation (comparing type D with type C) reduces the annual heat losses by 28%, with the largest differences occurring during the winter period. The deviation depends mainly on decreasing the thermal bridge towards the environment above the soil. The heat losses towards the soil under the foundation are the same in both cases.

Concerning the maximum rates of heat losses of the different cases (from simulations with snow and soil freezing) (Table 4), the deviations are clearer. Comparing the uninsulated crawl space foundation (A) and the extra-insulated slab foundation (D), one can observe that with type D the maximum heat flow rate is reduced by 55%. Comparing the two crawl space types (A and B), one finds that with type B the maximum heat flow rate is reduced by 44%, and the uninsulated crawl space foundation (A) has a maximum heat transfer rate which is 1.8 times higher than that of type B. Comparing foundation type C and D, one finds that the edge insulation in type D reduces the maximum heat transfer rate by 39%. From Figure 10 it can be seen that type D has the lowest variation over the year, which is because it is more insulated against the vertical sides compared to the other foundation types. For all the investigated foundation types, insulation at the vertical sides has a major impact on the heat losses and should always be installed.
4.6 Soil temperature

In Figure 12 the temperature distribution in the soil adjacent to a foundation of the uninsulated crawl space type (A) is presented for four different days during the year. The days represent the different annual seasons of spring, summer, autumn and winter. During the summer (8-AUG in Figure 12) the soil temperature increases and is heated from above (by the air) due to the large thermal inertia of the soil. The temperature distribution in the horizontal direction is even since the heat is fed from the air, and the difference in temperature between the outdoor air and the crawl space is lowest during the summer. During the autumn (20-NOV) the heat transfer direction is reversed, heat flows from the soil to the colder air and the soil temperature decreases. Under the foundation the heat flux is smaller, so the soil temperature does not decrease at the same rate. During the winter (22-JAN) the soil outside the foundation exhibits a frost penetration depth, which can be seen as the dark blue area in Figure 12; beneath the foundation the cooling is less intensive and isol therm curves are created. The frost penetration depth is 0.45 m at the edge of the simulation volume and horizontal, which indicates that the simulation volume, with 5 m outside the foundation, is sufficient to predict the temperature changes in the soil. Under the foundation the temperature never decreases below freezing point during a whole year and frost heave is therefore avoided. The insulation from snow and the soil freezing increase the thermal resistance between the air and soil. The heat stored in the deep soil layers prevents the temperature from decreasing to lower values. During the spring (8-MAY), when the snow has melted, the soil is heated by the air. During the winter, the soil temperature has been decreasing down to a depth of around 8 m. An area with a lower temperature at some depth down in the soil appears due to the inertia of the frozen soil. This region is heated from the
top and from beneath until it finally disappears. At a depth of around 8 m, the temperature in the soil is almost constant over the year.

**Figure 12.** Temperature distribution in the soil during the different seasons for foundation type A.

For the other foundation types, the soil temperature at the sides of the simulation model will have the same shape as in Figure 12 over the year. For the area under the foundation, the yearly variations will be smaller, since foundation type B-D are insulated downwards, which causes less heat exchange between the foundation and the soil. Heated regions like those in Figure 13 will appear for case C-D during the whole year, because of the higher temperatures in the concrete compared to those in the crawl space, which has a system of joists between the indoor floor and the air in the crawl space.

In Figure 13 the soil temperatures for the different foundation types on 22nd January are presented for comparison. Concerning the area below the foundation, it can be seen that the colder temperature in the crawl space volume of foundation type A (compared to that of type B) causes the lowest soil temperature below the foundation. Heat is transported to the crawl space from the soil. For type B the crawl space temperature is higher than the soil temperature and, together with the insulation layer, the soil temperature just below the foundation is of the same order as the deep soil temperature. For case C and D the downward insulation has the same thickness and, therefore, the temperatures in the soil under the foundation have the same profile for these cases. The difference between these types is that the sideward rate of heat losses is higher for case C.
In Table 5 the rates of heat losses downward and sideward for the different foundations are presented for 22nd January. The downward rate of heat losses to the soil is highest for case C and D, causing the highest soil temperatures under the foundations. Case B has a heat flow rate downwards to the soil of -50 W and for type A there is a positive heat flow rate from the soil to the foundation of +157 W. Concerning the foundations’ vertical sides, type A has the highest rate of heat losses, and type B and C have approximately the same sideward heat flow rates, while case D shows a heat flow rate of only 25 W. Overall, the sideward heat flow rates have a large influence on the total amount of heat losses during the winter. On 22nd January the sideward heat flow rate was for case A-C ≥ 50% of the total heat flow rate, and for case D, with more insulation at the vertical sides to prevent cold bridges, the sideward heat flow rate was only 18% of the total heat flow rate.

Table 5. Relationship between the downward and sideward heat flow rates on 22nd January.

<table>
<thead>
<tr>
<th>Foundation type</th>
<th>Downward heat flow rate (W)</th>
<th>Sideward heat flow rate (W)</th>
<th>Sideward heat flow rate (% of total)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>+157</td>
<td>-465</td>
<td>151%</td>
</tr>
<tr>
<td>B</td>
<td>-50</td>
<td>-121</td>
<td>71%</td>
</tr>
<tr>
<td>C</td>
<td>-113</td>
<td>-114</td>
<td>50%</td>
</tr>
<tr>
<td>D</td>
<td>-113</td>
<td>-25</td>
<td>18%</td>
</tr>
</tbody>
</table>

Figure 13. Winter temperatures in the soil for the different foundation types (on 22nd January).
4.7 Risk of frost heave and improvement with frost protection insulation

Table 6 presents the lowest temperatures beneath the foundation for the different types. For type A-D, the results are presented for the total model and a model without a snow layer. For type C and D, the results are presented for models representing worst-case scenarios: one model without frost protection outside the foundation and one model without any snow layer and without frost protection insulation outside the foundation. With a snow layer included, all the foundation types have temperatures above the freezing point of water beneath the foundation, but for the cases with a snow-free area around the foundation, all the foundation types exhibit temperatures below zero under the foundation. The frost protection insulation outside the foundation results in an increased temperature below the foundation for the coldest spots, but without snow layer and soil freezing, the temperature never rises above 0°C below the foundation. The values in the first column of Table 6 (for the total model, including both snow and the freezing of water in the soil) are probably the temperatures closest to the real values under the foundations.

Table 6. Lowest temperatures beneath the foundations.

<table>
<thead>
<tr>
<th>Foundation type</th>
<th>Total model (°C)</th>
<th>Without snow in model (°C)</th>
<th>Total model without frost protection outside foundation (°C)</th>
<th>Without snow and without frost protection outside foundation in model (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>1.12</td>
<td>-3.26 (1.43m)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>B</td>
<td>1.60</td>
<td>-2.94 (1.67m)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>C</td>
<td>2.66</td>
<td>-1.84 (0.85m)</td>
<td>1.67</td>
<td>-9.18 (1.23m)</td>
</tr>
<tr>
<td>D</td>
<td>1.78</td>
<td>-3.24 (0.94m)</td>
<td>0.85</td>
<td>-9.12 (1.03m)</td>
</tr>
</tbody>
</table>

The results for the heat flux from the simulations of foundation type C and D with frost protection outside the foundation and the corresponding results from the simulations of type C and D without frost protection are presented in Figure 14 for comparison. With frost protection, a small decrease in the heat flux from the foundations during the winter (grey lines in Figure 14) can be observed, compared to the heat flux without frost protection (blue and red dashed lines). On an annual basis, the heat losses were almost the same for both cases, but with frost protection insulation, small improvements occurred during the winter and the opposite occurred during the summer.

Overall, the results show that snow reduces the risk of frost heave for all the foundations. With snow included, there is no risk of frost heave for this soil setting. Frost protection
insulation increases the temperature under the foundation and decreases the frost penetration depth, thereby reducing the depth of the landfill needed. As shown earlier, the extra insulation has a minor impact on the heat losses from the foundation.

Figure 14. Heat fluxes with and without frost protection insulation: a) foundation type C, b) foundation type D.

4.8 Passive house standard

Including snow and soil freezing reduces the heat losses through the foundation, and the requirements for the other building surfaces are therefore lowered with regard to complying with the passive house standard. All the foundation types complied with the limit for specific energy according to the standard.

If snow and ground freezing are not considered, foundation type A does not fulfil the criteria of the standard.

The annual heat demand for a one-storey passive house for all the foundation types is 2-3% higher without snow and ground freezing. The 1D simulation based on the ISO 13370 standard overestimates the annual heat demand by around 5% for foundation type A and B. The annual heat demand is, however, not affected substantially by the overestimation.

For classifications made according to the passive house standard, the heat loss factor must also be considered. The heat loss factor is the dimensioning variable, since this criterion is usually more difficult to fulfil compared to the specific energy criterion. Figure 15 shows the requirement for the average U-value of the rest of the building envelope with regard to
fulfilling the heat loss factor criterion; the red dotted lines represent typical average U-values for a passive house. From Figure 15a it can be seen that foundation type A and C are not suitable for a passive house in a subarctic climate. It is probably not possible to use foundation type B without including a snow layer in the calculations if one is to achieve compliance with the passive house standard. It can also be seen that snow and soil freezing have a large impact on foundation constructions with less insulation on the vertical sides of the structure, such as foundation A, B and C. In Figure 15b the corresponding impact is smaller, but still distinct.

When using the ISO 13370 standard to simulate the annual heat losses, the error caused by excluding snow is often reasonably small, but the difference in the heat loss factor is not negligible for regions with snow during a significant part of the year. If snow and ground freezing are considered in the building simulation models, more buildings in subarctic regions will fulfil the requirement for the passive house standard, since the heat loss factor is often the dimensioning parameter.

**Figure 14.** Requirement for the average U-value for the rest of the building envelope (excluding the foundation) with regard to complying with the passive house standard: a) one-storey house, b) two-storey house.
5 Conclusions

It is possible to achieve a good thermal estimation of the temperature levels in a crawl space foundation for a whole year with a numerical heat model if soil freezing and snow layer predictions are included. Daily based time steps are sufficient in order to simulate the heat losses from a foundation.

Snow is an important parameter for correct determination of the heat losses of foundations in a subarctic climate, while soil freezing only affects the heat losses to a minor extent. The reduction in the annual heat losses through the investigated foundations varies from 7-10%, and the investigated foundation types are those most commonly used in Sweden today; the maximum heat loss rate was determined to be 12-25% for the studied year. The present study shows that for all the investigated foundation types, insulation at the vertical sides has a major impact on the heat losses and should always be installed.

The 3D model implemented in this study can be used to investigate the risk of frost heave affecting buildings. This paper shows that a snow layer and soil freezing have a significant effect on the temperatures beneath the foundation.

Frost protection insulation outside the foundation should be installed in order to avoid frost heave and reduce the amount of landfill in a subarctic climate.

For passive houses, calculations of heat losses through the foundation should include snow as a factor, since the requirements for this construction type are difficult to fulfil.

The commonly used calculation methods for heat transfer through the foundation, based on the standard ISO 13770 (using a 1D model), are thorough enough for climates without snow and frost. In subarctic climates, however, a more comprehensive approach is needed to predict the heat transfer fully.

References

3. ISO 13791 (2004). Thermal performance of buildings - Calculation of internal temperatures of a room in summer without mechanical cooling General criteria and validation procedures
14. Johansen O. 1975 Thermal Conductivity of Soils, Draft Translation Report 637, Cold Region Research and Engineering Laboratory, Hanover, NH, USA.


18. SMHI weather station data


