3D HEAT TRANSFER ANALYSIS IN ARCHITECTURAL MODELING: A CASE STUDY WITH OPENFOAM

A Dissertation
Presented to
The Academic Faculty

By

Maryam Almaian

In Partial Fulfillment
of the Requirements for the Degree
Masters of Science in the
School of Architecture
College of Design

Georgia Institute of Technology

April 2024

3D HEAT TRANSFER ANALYSIS IN ARCHITECTURAL MODELING: A CASE STUDY WITH OPENFOAM

Thesis committee:

Dr. Patrick Kastner
School of Architecture
Georgia Institute of Technology

Dr. Tarek Rakha
School of Architecture
Georgia Institute of Technology

Tyrone Marshall
Co-Director of the Energy Lab
Perkins & Will Research

Date approved: April 24, 2024

For my parents, Njoud & Talal Almaian

ACKNOWLEDGMENTS

I want to express my deepest gratitude to my advisor, Professor Patrick Kastner, for his exceptional guidance, constant support, and mentorship throughout my academic journey and master's thesis. His dedication to excellence and passion for teaching me OpenFOAM has been truly inspiring. Professor Kastner always pushed me to be the best, challenging me to exceed my expectations and strive for excellence in every aspect of my work.

When I approached Professor Kastner with the idea of pursuing an independent study in OpenFOAM, despite my initial lack of knowledge in the field, he was incredibly encouraging and supportive. Professor Kastner not only motivated me to undertake the project but also provided comprehensive guidance and instruction, patiently teaching me the intricacies of OpenFOAM and equipping me with the necessary skills to succeed.

I am also grateful to Professor Tarek Rakha for his generosity with time and resources. The amount of knowledge and skills I learned from him is extensive. Finally, his commitment to my academic and personal growth has made a significant impact on my journey. Furthermore, I would like to extend my appreciation to Tyrone Marshall my instructor and the co-Director of Energy Lab Perkins&Will, whose skills have enriched my academic journey in various ways. I would also like to thank Robin Tucker for always being there when in need of assistance. Additionally, I am thankful to my colleagues at the High-Performance Lab for their continual support, sharing of knowledge, and invaluable tips that contributed to my success; Anuradha Kadam, Tarek Sherif, Rawad Elkontar, and Yasser Elmasri.

Lastly, I am deeply thankful to my parents, aunts; Fatoma, May, and Fay, siblings; Hamad, Muneera, Ali, Yousef, and my best friend Noof for their love, encouragement, and understanding throughout this stressful time. Their unconditional support has been my source of strength and motivation. I am sincerely grateful to each and every one of the individuals who helped me reach this step in my academic work.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

SUMMARY

As the global focus on sustainable building practices intensifies, architects face the challenge of designing structures that meet certain aesthetic and functional criteria while minimizing energy consumption. One critical aspect of achieving energy-efficient buildings is the selection of appropriate building materials with optimal thermal properties.

The tools and software to simulate 2D heat transfer are available but often limited in their set of features and / or cost prohibitive. **1.Add sentence of what cannot be done with 2D analysis? Complex geometry? Corners? Full building envelope analysis?** The integration of 3D thermal performance analysis into the architectural design process is an even more complex and underdeveloped area.

This thesis aims to address this gap by exploring the use of *OpenFOAM* to develop a user-friendly tool to simulate building-related heat transfer problems. The outcomes of this thesis aim to empower architects to make informed decisions about material selection, their impact on energy efficiency, by seamlessly embedding it into the Rhino & Grasshopper CAD environment.

CHAPTER 1
INTRODUCTION AND BACKGROUND

Architects and engineers refer to ASHRAE standards to comply with the minimum required insulation based on the location of a project. In adhering to these standards, there is the potential to optimize the selection of materials toward lower energy demand and to increase thermal comfort. However, modeling 3D heat transfer for applications in buildings is a complex and long process for modelers due to the lack of available free software.

This thesis aims to bridge the gap between providing architects with easy-to-use 3D heat transfer software that is integrated into architect design software, such as *Rhino & Grasshopper*. Here, we used *OpenFOAM (OF) / Grasshopper* to construct an envelope segment from *ISO 10211:2007* [1], then calculate heat transfer, and assess whether the validated case complies with our results.

2.You need a section here that describes the problem statement AND the research questions your thesis answers. Please check my PhD thesis or masters thesis for an example of how to do this.

CHAPTER 2

HEAT TRANSFER IN BUILDINGS

The capabilities of current software packages are specific to remodeling one wall or section in separate software, rather than integrating it into the design process using design software packages. **3.name the software you are talking about. Therm?** This requires the user to remodel walls, sections, or entire buildings, depending on the scale of the project and the number of design changes involved.

Traditional building design processes often rely on simplified 2D heat transfer models, which fail to capture the precise thermal interactions that occur in the real world. This limitation affects the accuracy of performance predictions and affects the identification of optimal design solutions for energy efficiency. 3D simulations offer a more realistic representation of thermal transfer, accounting for both conduction and convection. Moreover, they hold massive potential for cost reduction and carbon emission mitigation. By accurately predicting thermal loads and optimizing HVAC systems**4.you are not doing HVAC here, remove that**, designers can minimize energy consumption, thus reducing operational costs for building owners.

2.1 Literature Review

Current tools are limited to 2D heat transfer, such as *HTflux* [2], which is specialized software for simulations of two-dimensional heat and water vapor transport. **5.add table of figure that shows overview of all tools you found: HTflux, Therm, etc.** It uses the 2D Glaser method [3] to calculate dew points, condensation, and evaporation in 2D structures. The software offers an easy-to-use interface, can import CAD geometries, and employs a direct mapping method for accurate simulations. It provides various thermal analysis metrics (heat flow, U-value, ψ-value, etc.) and measures extreme temperature values. Two contributions stand out that discuss different 3D heat transfer elements and approaches by [4], and [5], whereas one contribution presents 2D heat transfer using OpenFOAM [6].

[6] presented a case study that presents a thermal bridge analysis and 2D heat transfer with OpenFOAM, which is similar to our approach due to the use of OF. The simulation is constructed from pre-processing, simulation, and post-processing. Based on the result of the used case, it is possible to use *Rhino & Grasshopper* to calculate architectural heat transfer in the pre- and post-processing phases, but this requires additional research to achieve this goal. The authors experimented with a validation case retrieved from *HTflux* that was originally published in the ISO guide [6, 1].

The second approach is found from [4] which experimented with 3D heat transfer following a different method, which is to develop a mesh using lumped hexahedral elements and employs ray/triangle intersection techniques for an accurate geometric representation of the building. It applies an energy balance to each element and integrates a system of ordinary differential equations to obtain spatio-temporal indoor temperature and relative humidity fields. However, there are several limitations, such as not explicitly solving flow fields, which may limit its accuracy in capturing complex airflow patterns. Furthermore, idealized thermal conditions do not include real world conditions and do not accurately represent the impact of the thermal mass of the floor. [4].

The third approach by [5] used *COMSOL / Multiphysics* to simulate heat transfer in buildings and has been validated. There were minor differences in the results due to the convective heat transfer coefficient. The main issues with the software are the cost and the need to use software that is not integrated in the design software. However, the authors' work is valuable in terms of offering heat transfer simulation focusing on buildings.

The final research by Zhong et al., 2019 [7] is directly connected to this research workflow of investigating the computational fluid dynamics (CFD) simulation of convective heat transfer but specifically on Japanese vernacular architecture, focusing on ”machiya” buildings. These traditional structures offer passive strategies for maintaining indoor comfort with minimal HVAC assistance. The study aims to validate and develop a methodology for simulating convective heat transfer using high-resolution 3D steady Reynolds-averaged Navier–Stokes (RANS) CFD simulations. Through comparison and evaluation of different RANS models and boundary layer modeling approaches, the researchers identify suitable methods for predicting convective heat transfer coefficients (CHTC) and flow fields. Validation against wind-tunnel experiments on heated cubes in turbulent channel flow confirms the effectiveness of selected models and approaches. The study contributes to advancing simulation methodologies for sustainable building design but did not meet the objective of automating the heat transfer workflow to simplify it for the user [7].

2.2 Heat Transfer

6.needs better name for section

This thesis presents a simulation based on conductive and convective heat transfer. One may calculate conductive heat transfer using the heat diffusion equation [8]:

$$\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} + \frac{\dot{q}}{k} = \frac{1}{\alpha}\frac{\partial T}{\partial t}, \text{ with } \alpha = \frac{k}{\rho c_p} \tag{2.1}$$

Here, x, y, and z are the Cartesian coordinates, α is the thermal diffusivity, $\dot{q}$ is energy generation rate per unit volume, and c_p is the specific heat capacity.

To calculate convective heat transfer, the used formula accounts for both advection (the movement of heat with the fluid flow) and diffusion (the spatial variation of temperature):

$$\frac{\partial T}{\partial t} + \mathbf{v} \cdot \nabla T = \alpha \nabla^2 T + \frac{\dot{q}}{k}, \tag{2.2}$$

where $\mathbf{v}$ is the velocity vector of the fluid, α is the thermal diffusivity, ∇ stands for the gradient operator, ∇^2 represents the Laplacian operator, $\dot{q}$ is the energy generation rate per unit volume, and k is the thermal conductivity [8].

7.add sentence on conjugate heat transfer approach that tries to offer an approach to solving such problems, see Wikipedia.

8.Add sentences of how this cannot be done by hand due to the nature of the differential equations.

9.I would actually talk about the history first, then the equations. I reordered it now.

2.3 Computational Fluid Dynamics

The approach this research follows uses OpenFOAM which stands for Open-source Field Operation and Manipulation and is a widely used open-source computational fluid dynamics (CFD) software package that allows engineers and researchers to simulate fluid flow, heat transfer, and other related phenomena using numerical methods. The project uses CFD to calculate the heat flow gradient in the space.

10.add section on CHT solver that we are using for this.

CHAPTER 3

2D HEAT TRANSFER METHODOLOGY

The first step before experimenting with the 3D heat transfer simulation is to understand the simplest form of heat transfer in buildings. This chapter presents the calculation of a 2D section of a single-layer brick wall. Followed by the experimental design, results, and discussion.

3.1 Manual Estimation of Heat Flux

The heat flux calculation for the 2D section shown in Figure 3.4 **(a)** is as follows:

$$q = -k\frac{dT}{dx} \tag{3.1}$$

where q is the heat flux, k is the Thermal conductivity, and $\frac{dT}{dx}$ is the Temperature gradient, which can be found by $\frac{T_2 - T_1}{L}$ over thickness L [9].

Solving for q where k = 1 W/m^2, L = 0.43 m, T_1 = 25.8 $^\circ$C, T_2 = 21.1 $^\circ$C

Substituting the given values:

$$q = -1\,\text{W/m}^2 \times \frac{21.1\,^\circ\text{C} - 25.8\,^\circ\text{C}}{0.43\,\text{m}}$$

$$q = 10.93\,\text{W/m}^2$$

So, the heat flux q is $10.93\,\text{W/m}^2$.

3.2 Experiment

11.needs better section title, not just "experiment"

This section presents a real-time measurement of the 2D brick wall with a duration of

74 hours 12.add dates and which building. where did we measure using U-value and heat flux sensors as shown in Figure 3.1 **(a)** and Figure 3.1 **(b)**. Then followed by a 2D simulation section where the same experimental design conditions are implemented in HTflux [2] a 2D heat transfer simulation software.

(a) **(b)**

Figure 3.1: Outdoor heat flux sensors during experimental design. The window was closed as much as possible to avoid interference with the sensor cord **(a)**. Indoor heat flux sensors during the experimental design **(b)**.

Experimental Design

The measurement device used was the gSKIN® KIT-2615C (calibrated) U-Value and Heat Flux Measurement Kit from GreenTeg [10] shown in Figure 3.2. The schematic in fig. 3.4 shows the experimental design setup with the same conditions of k = 1 W/m^2 L = 0.43 m, T_1 = 25.8 °C, T_2 = 21.1 °C .

Figure 3.2: The U-Value and Heat Flux Measurement Kit, for details see table A.1.

Experiment Results

The chart in 13 represents the reading results of 74 hours. The final resulting U value = 2.31 W/m^2k from the report and the temperatures show compliance with the calculation section above q is $10.93\,W/m^2$. Where U-value from the report = 2.31 W/m^2k and

$$U = \frac{k}{\text{L}} = \frac{1}{0.43} = 2.33W/m^2k \tag{3.2}$$

Where k = 1 W/(m²), L (thickness) = 0.43. So, U_{calc} = U_{exp} = 2.31 W/m^2k

13 below is a plot report showing the experimental design results of the fluctuations in indoor and outdoor temperature along with the heat flux. From the report, three temperatures from different time steps were selected to be compared with the 3D simulation results. Also, Table 3.1 shows the comparison of the selected three points, where T_{val} is the experiment's resulting temperature and T_{sim} is the simulation's resulting temperature.

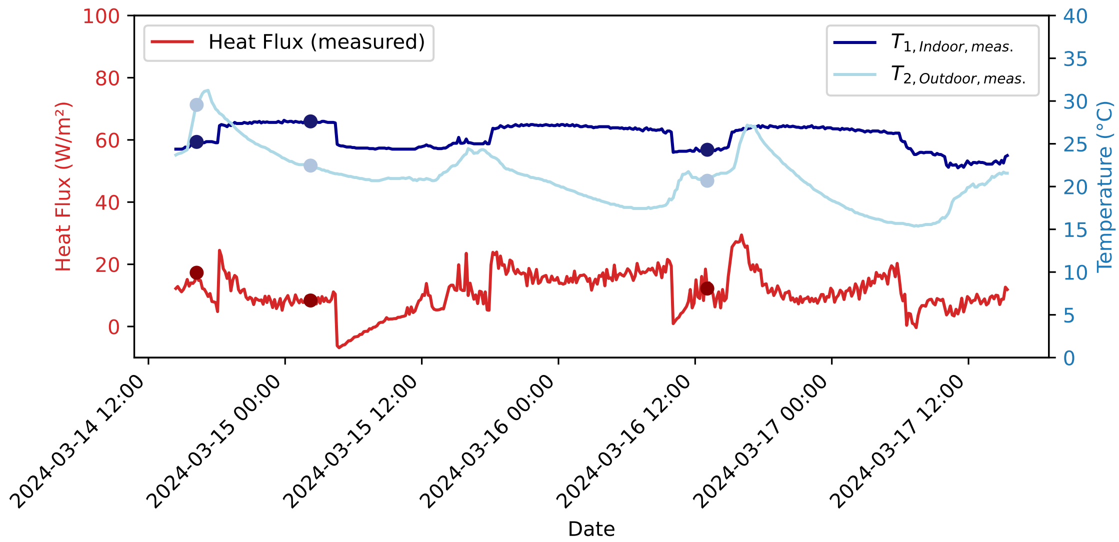


Figure 3.3: The heat flux results over time.

13.needs more detailed caption

Table 3.1: 2D Temperature and Heat Flux Comparison

Time	T1val_in	T1sim_in	T2val_out	T2sim_out	Qval	Qsim
3/14/2024 16:14	298.39	298.37	302.7	302.7	17.38	17.29
3/15/2024 02:14	300.76	300.768	295.51	295.6	8.99	8.36
3/16/2024 13:04	297.4525	297.452	293.827	293.827	12.54	12.2

14.make sure to have reasonable significant decimals everywhere. stay consistent. 15.Make sure to format variables correctly, such as $T_{1,val,in}$

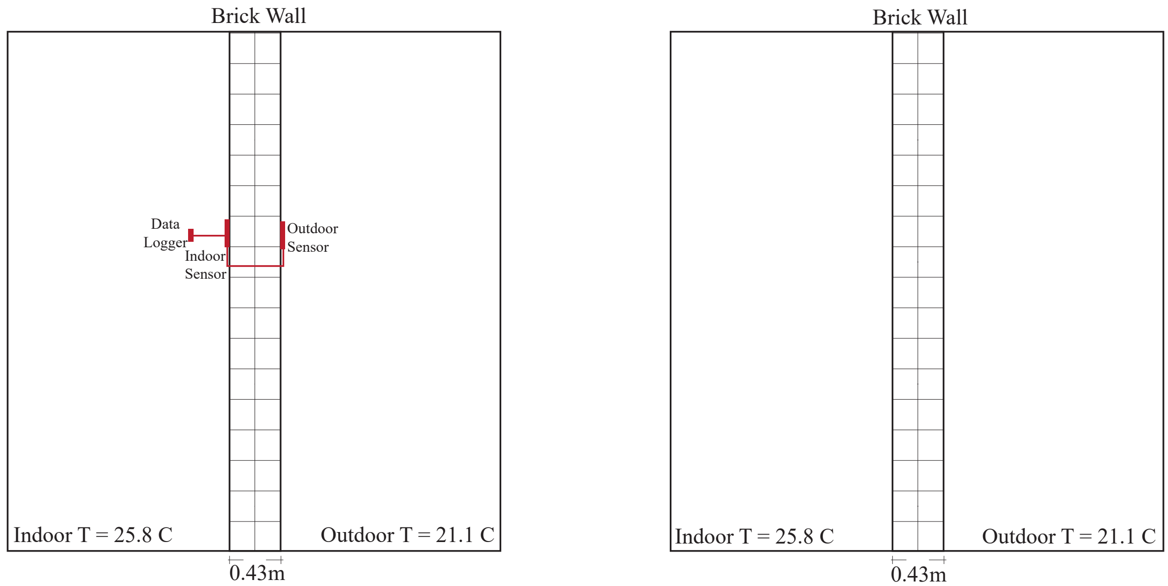
3.3 Simulation

17.add subsections here that make very clear that the first simulations are done with OpenFOAM, the others are done with HTFlux

After successfully finding and ensuring the compliance of the heat flux in the two methods, which are the calculation and the experiment, the final method uses a 2D heat transfer

Table 3.2: 2D Results Percentage of error

Metric	T1 Percentage Error	T2 Percentage Error
Average	0.0032%	0.0102%
Standard Deviation	0.0033%	0.0176%



(a) Experiment setup for the 2D brick wall.

(b) Section of the 2D brick wall

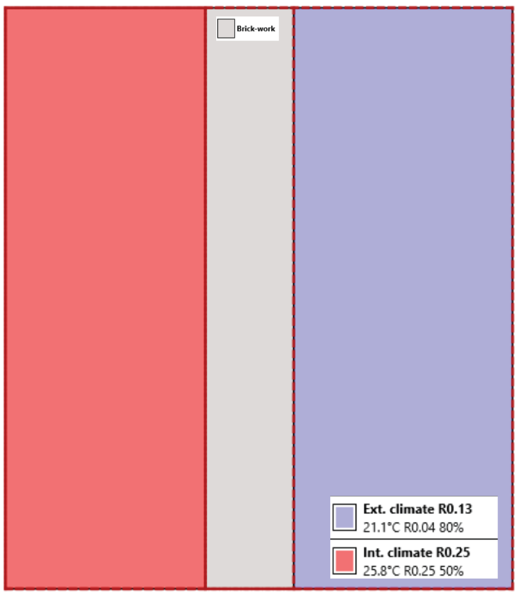
Figure 3.4: 2D Section and Setup

16.figure a is probably enough here

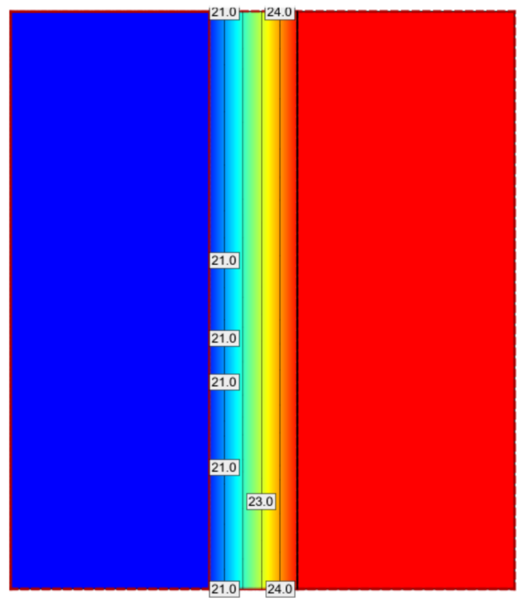
simulation, HTFlux [2]. The same brick wall with the same boundary conditions is constructed in the software as shown in Figure 3.5 **(a)** which shows the materials and the boundary condition and **(b)** represents the 2D simulation results where the resulting heat flux = q is $10.91\,W/m^2$ as expected.

3.4 Discussion

This section successfully presented the resources to calculate the heat flux of a 2D brick wall using three methods, which are by calculation, using sensors, using 2D simulation software where the heat flux q, respectively, = $10.93\,W/m^2$, $10.93\,W/m^2$, and $10.91\,W/m^2$. However, the gap of 3D heat transfer simulation is still missing. Thus, chapter 3 showcases the workflow of a free 3D heat transfer simulation.



(a) The simulation boundary conditions from HTFlux



(b) The brick wall temperature gradient results from HTFlux

Figure 3.5: 2D HTFlux Boundary conditions and Results

18.add heat flux you found also in figure caption.

CHAPTER 4

3D HEAT TRANSFER METHODOLOGY

19.reiterate why 3D simulations might be advantageous

The methodology of the simulation begins with the validation case retrieved from [1] is a 3D geometry consisting of the first and second floors separated by the floor slab and the plaster floor, the walls are aerated concrete, insulation, and brick. **20.sentence doesn't sound right** Below is a description of each component in the design software section and the roles of each software used Figure 4.2 **(a)**. **21.below where?**

22.this transition makes no sense Then, we use Rhinoceros to create, export, and import geometries to create the mesh. *Rhinoceros* will be used as a user interface to model the geometry and connect it to Grasshopper to calculate 3D heat transfer. Grasshopper (GH) is a visual programming environment that we use to automate the OF simulation. Automation can find the coordinates of the points in the region to set the mesh, boundary conditions, and material properties. In addition, we use *OpenFOAM*, which is an open source computational fluid dynamics (CFD) software package that is used to simulate fluid flow, heat transfer, and other capabilities.

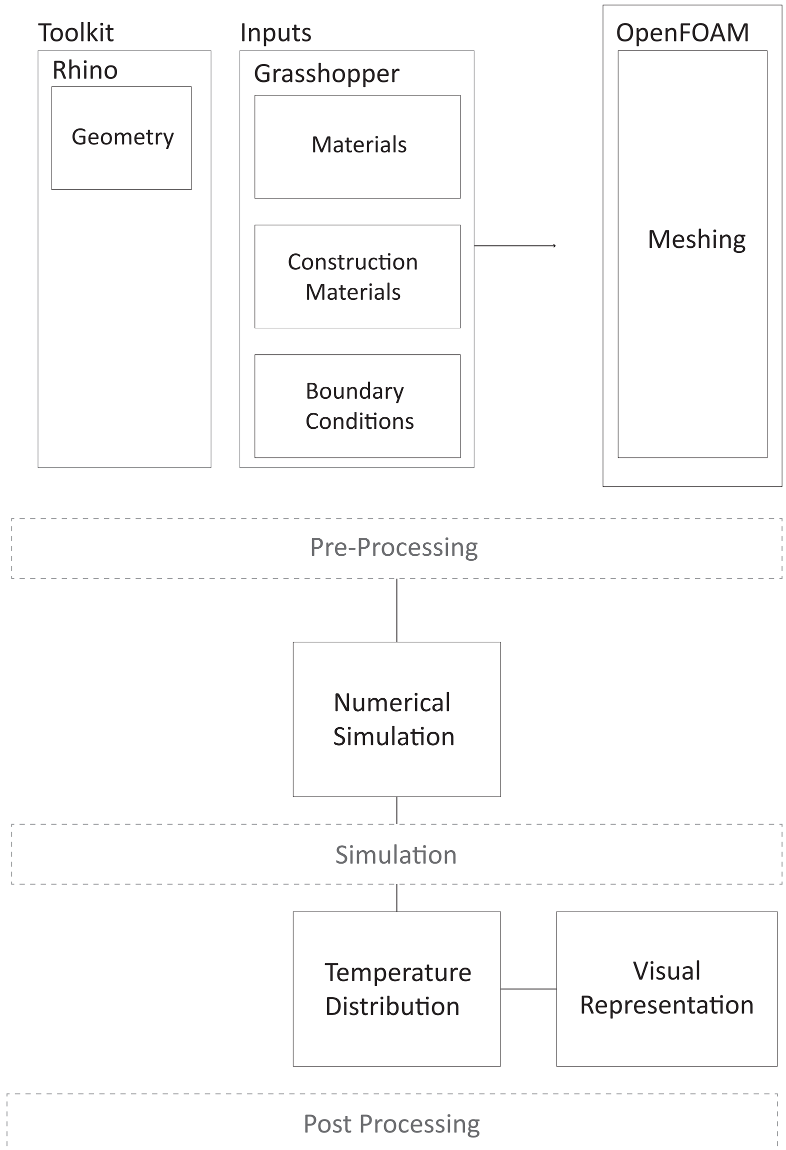


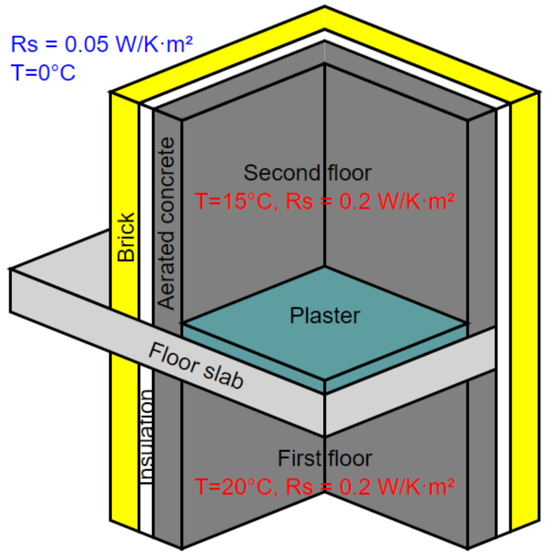
Figure 4.1: Flowchart of simulation steps starting from pre-processing to the post-processing of the simulation.

4.1 Validation Case Study

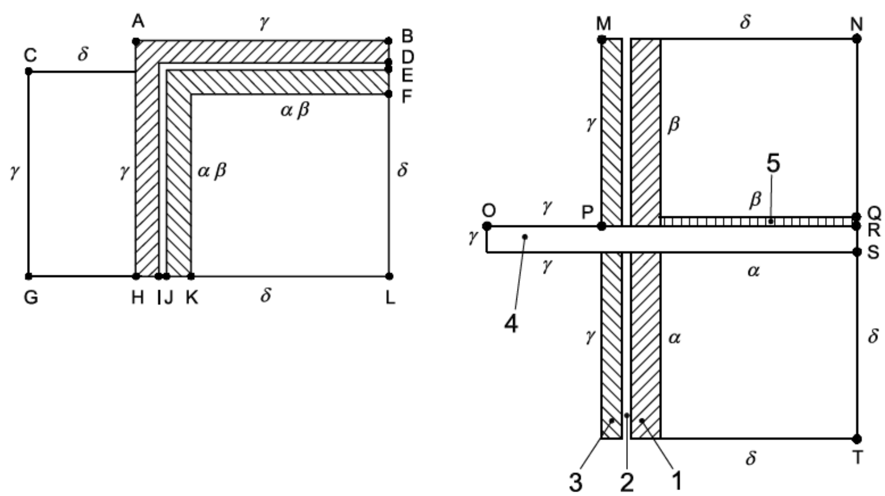
The case study used for this project is a validated 3D heat transfer case documented in *ISO 10211:2007*[1] [1]. Beyond *ISO 10211:2007*, it is also documented in the *QuickField* software where the properties, layers, and boundary conditions of the materials are accessible,

[1]ISO 10211:2007—Thermal bridges in building construction. Validation of case A.3

which we exported to compare it with our method.



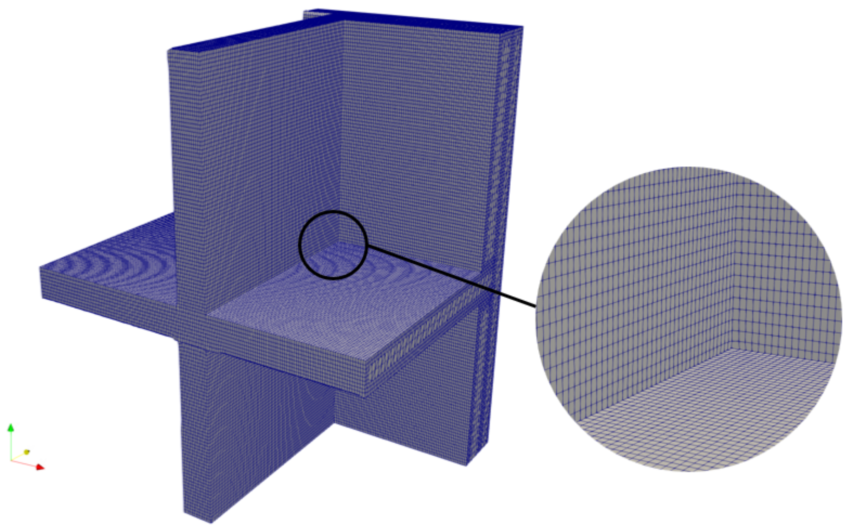
(a)



(b)

Figure 4.2: Validation case materials [1] **(a)** and Validation case sections retrieved from BS EN ISO 10211:2007(E) **(b)** [1].

QuickField's Heat Transfer module offers versatile features including steady-state or transient formulations with customizable initial field distributions and flexible time parameters, accommodating nonlinear specific heat and nonlinear or anisotropic properties. Although, the results include diverse thermal field mappings such as temperature, heat flux, and thermal gradients, editing and customizing the post-process options and locations were limited.



(a)

Type	Multi-block dataset
# of Cells	360,364
# of Points	3,481,036
# of TimeSteps	400
Bounds X	0 to 1.9 (delta: 1.9)
Bounds Y	0 to 1.25 (delta: 1.25)
Bounds Z	-1 to 1.15 (delta: 2.15)

(b) 3D Validation case mesh statistics.

Figure 4.3: **(a)** *OpenFOAM* mesh viewed in $ParaView$ **(b)** Validation case mesh statistics retrieved from BS EN ISO 10211:2007(E) [1].

4.2 Pre-processing

The pre-processing phase consists of dividing the geometry into different zones based on different materials and locations. In addition, thermophysical properties, such as specific heat capacity and thermal conductivity, are assigned to materials, and fluid or solid properties are assigned to regions. Limit conditions and construction materials are specified in fig. 4.2 **(a)** and **(b)**.

Table 4.1: Construction material properties and boundary conditions used for the simulation domain. Data were taken from the demo example in *QuickField*.

Materials	$k\ \left[\frac{\mathrm{W}}{\mathrm{m\,K}}\right]$	$c_p\ \left[\frac{\mathrm{J}}{\mathrm{kg\,K}}\right]$	$\rho\ \left[\frac{\mathrm{kg}}{\mathrm{m}^3}\right]$
Floor Slab	2.5	1000	2300
Aerated Concrete	2.5	1000	2300
Brick	0.7	1060	710
Insulation	1	1450	35
Plaster	1	1000	2300

Table 4.2: 3D Boundary Conditions

Boundary conditions	$T[°\mathrm{C}]$	BC type
Inside temp. (1st floor)	20	fixedValue
Inside temp. (2nd floor)	15	fixedValue
Outside temp.	0	fixedValue

Case Study setup

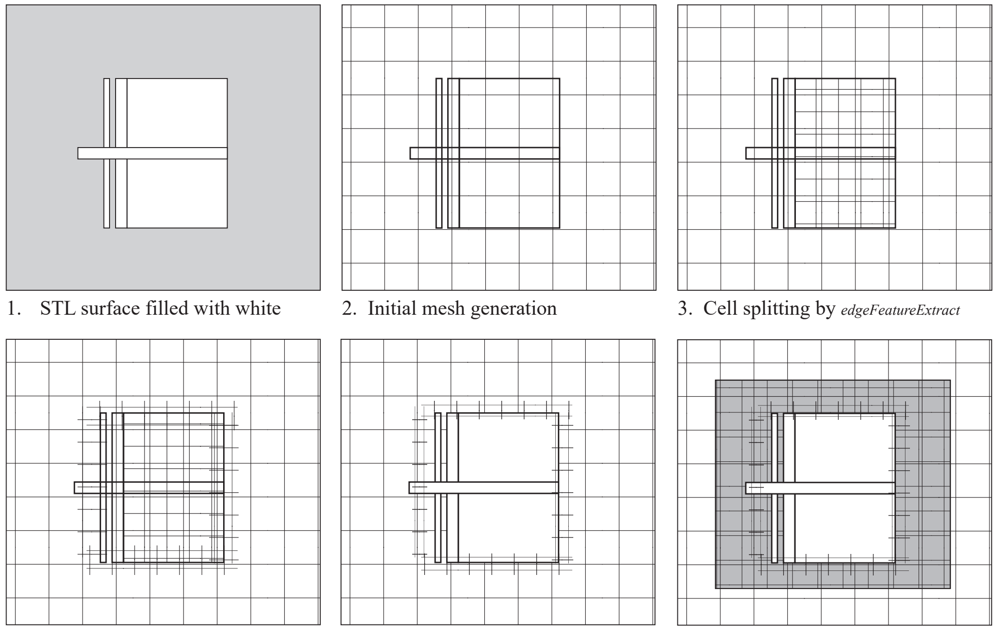
The case study geometry was exported from *QuickField* and then modeled in *Rhinoceros* and subsequently exported as a mesh to be processed with OpenFOAM where Figure 4.4 visualizes the steps to create the mesh. The validation case was constructed in *OpenFOAM* by creating the mesh and *snappyHexMesh* files. Figure 4.2 (b) illustrates the *OpenFOAM* mesh visualized in *ParaView*.

4.3 Simulation Equations

4.3.1 Solver Setup

The solver used in this case is *chtMultiRegionFoam* in OpenFOAM 2306 which is a solver capable of solving for steady or transient fluid flow with solid heat conduction and conjugate heat transfer between regions, buoyancy effects, turbulence, reactions, and radiation modeling[2] [11]. There are three solvers capable of simulating steady-state heat transfer between fluids and solids which are *chtMultiRegionFoam*, *chtMultiRegionSimpleFoam*, and

[2] Not used in this study.



1. STL surface filled with white
2. Initial mesh generation
3. Cell splitting by *edgeFeatureExtract*
4. Cell splitting by *surfaceFeatureExtract*
5. Cell removal
6. Cell splitting by region to create the zones using *splitMeshRegions*

Figure 4.4: Visualization of the steps and executables to create the mesh step-by-step

23.nice figure, but I am not sure it shows what is happening. Aren't we cutting the cells outside of the geometry, leaving the inside? This is the opposite of what Eddy3D does.

chtMultiRegionTwoPhaseEulerFoam, but, the main difference is that the selected solver for this case is capable of doing both steady and transient states to have a more sufficient software. **24.more sufficient software?**

To precisely mimic real-world situations, The use of *chtMultiRegionFoam* solver is essential to have accurate 3D heat transfer results due to the availability of using fluid and solid. Figure 4.12 visualizes the capabilities of the solver with the various domains and interfaces of different temperatures, solids, and fluids. A description of the boundary conditions can be found in section 4.2. However, the outdoor temperature is set to be 0 degrees Celsius and climate Setup can be easier by using Grasshopper to leverage the initial temperatures based on the location. Another crucial aspect is identifying the thermal properties that will allow the solver to identify the thermal conductivity, density, and properties of the material and calculate the heat transfer accordingly.

4.3.2 Physical models

The physical models found in the constant file are required for the simulation to run according to the OpenFOAM documentation [12].

Turbulence Properties

The turbulence used for steady-state heat transfer is Reynolds Averaged Simulation (RAS), where the full form is anisotropic contribution of the Reynolds stress is shown below to influence the motion of a fluid, providing a comprehensive understanding of fluid flow dynamics. Where the heat conservation equation according to [13]

$$u_i \frac{\partial T}{\partial x_i} + \frac{\partial}{\partial x_i}(K_T \frac{\partial T}{\partial x_i}) = 0 \tag{4.1}$$

where u_i is the velocity component in the i-direction, T is the temperature, x_i is the spatial coordinate in the i-direction, K_T is the thermal conductivity coefficient.

Then the Navier-Stokes equation, which describes the conservation of momentum for a fluid:

$$\frac{\partial(\rho ui)}{\partial t} + \frac{\partial}{\partial x_j}(\rho uiu_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\tau ij + \tau t_{ij}) + \rho g_i \tag{4.2}$$

Where, ρ is the density of the fluid, u_i is the velocity component in the i-direction, t is time, x_i is the spatial coordinate in the i-direction, p represents pressure, τ_{ij} is the viscous stress tensor, $\tau_{t_{ij}}$ is the turbulent stress tensor, and g_i is the gravitational acceleration in the i-direction [11].

Thermophysical Models

The thermophysical properties located in each zone and the constant file require inputs of density ρ, thermal conductivity, and specific heat capacity for every material.

Finite Volume Options

The fvOptions text file allows the user to further manipulate the systems of equations, such as sources and skins.

4.3.3 Solver Equations

25.are these the equations that CHT is based on? If yes, then say so. This section presents the implementation of heat transfer equations and metrics into OpenFOAM and the solver from OpenFOAM Foundations[14]

Fluid Equations

Mass Conservation is

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_j)}{\partial x_j} = 0 \tag{4.3}$$

Momentum Conservation Equation is

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_{rj} u_i) + \rho \epsilon_{ijk} \omega_i u_j = -\frac{\partial p_{rgh}}{\partial x_i} - \frac{\partial \rho g_j x_j}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\tau_{ij} + \tau_{t_{ij}}\right) \tag{4.4}$$

26.you already have a similar for of the NS equation above in the turbulence section

Solid Equations

Solid Energy Conservation is

$$\frac{\partial(\rho h)}{\partial t} = \frac{\partial}{\partial x_j}\left(\alpha \frac{\partial h}{\partial x_j}\right) \tag{4.5}$$

where h is the specific enthalpy, ρ is the density, and $\alpha = \frac{\kappa}{c_p}$ is the thermal diffusivity, and the specific heat capacity c_p.

Solid and Fluid Interface

$$T_f = T_s Q_f = -Q_s \kappa_f \frac{dT_f}{dn} = -\kappa_s \frac{dT_s}{dn} \quad (4.6)$$

where κ_f thermal conductivity of the fluid and κ_s the thermal conductivity of the solid.

4.3.4 Constructing The Case

An OF steady state heat transfer case requires three main folders to run which are iteration *0*, *constant*, and *system* folders. Each folder is responsible for properties or boundary conditions and includes these conditions in each zone in the geometry. The 10 zones in the case which are *Brick*, *Slab*, *Plaster*, *ConreteTop*, *ConreteBottom*, *IntAirTop*, *ExtAir*, *IntAirBottom*, *InsulationTop*, and *InsulationBottom*. Below in fig. 4.5 is an explanation of the case contents. The text files in this case are written automatically due to the script setup in GH. While constructing the constant file, the specific heat capacity of each material was needed, so we followed the steps below to find it (Brick Example):

$$\Delta T = 15°\text{C}$$

$$d = 100\,\text{mm} = 0.1\,\text{m}$$

$$A = 1300 \times (950 + 50 + 50 + 950)\,\text{mm}^2$$

$$A = 1300 \times (2000)\,\text{mm}^2 = 2.6\,\text{m}^2$$

$$Q = \frac{k \cdot A \cdot \Delta T}{d}$$

solve for the specific heat capacity (C_p):

$$C_p = \frac{Q \cdot d}{k \cdot A \cdot \Delta T}$$

$$Q = k \cdot A \cdot \frac{\Delta T}{d}$$

$$Q = 0.7\,\text{W/m.K} \times 2.6\,\text{m}^2 \times 15°\text{C}/0.1\,\text{m} \tag{4.7}$$

$$Q = 27\,\text{W}$$

$$C_p = \frac{27\,\text{W} \times 0.1\,\text{m}}{0.7\,\text{W/m.K} \times 2.6\,\text{m}^2 \times 15°\text{C}}$$

$$C_p = \frac{2.7\,\text{J}}{2.55\,\text{J/K}}$$

$$C_p \approx 1.06\,\text{kJ/kg.K}$$

So, the specific heat capacity of the brick is approximately $1.06\,\text{kJ/kg.K}$

To convert to J/kg.K:

$$C_p = 1.06\,\text{kJ/kg.K} \times 1000$$

$$C_p = 1060\,\text{J/kg.K}$$

27.add this to the appendix and refer to it in the text.

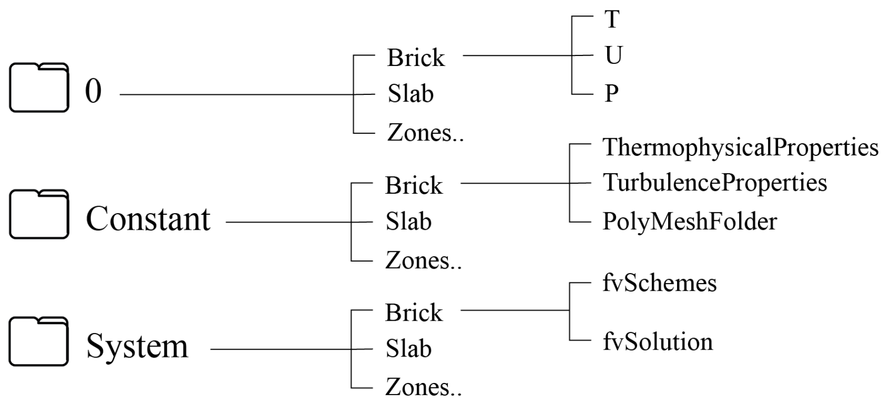


Figure 4.5: OF case folders to run. Where (zones..) represent a folder for each zone in the case.

4.3.5 OF Text files Automation by GH

This section explains the Grasshopper script that automatically creates and exports STL files, defines material properties, finds the point location in the meshes, and writes all the text files needed to run the OF case.

Materials Properties Component

The component shown in Figure 4.6 is a dictionary for the material connected to specific geometry. Each different zone in Rhino needs to be connected to the shown component by adding the corresponding specifications for the material such as the name, type, temperature, specific heat capacity, thermal conductivity, and density. This component is considered the base and many other sections in the script depend on it such as writing the other text files in 0 files where the boundary conditions are needed or writing the constant and system files.

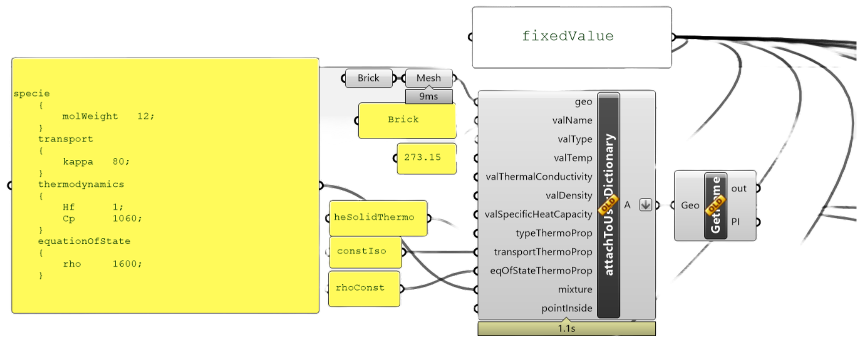


Figure 4.6: GH Material Properties Component

STL Files to Create the Mesh

The workflow shown in Figure 4.7 is used to create an STL from the geometry in Rhino and export it to the *trisurface* in the constant file to create the mesh using the mesh generation utility, *snappyHexMesh* where an explanation is in Figure 4.4.

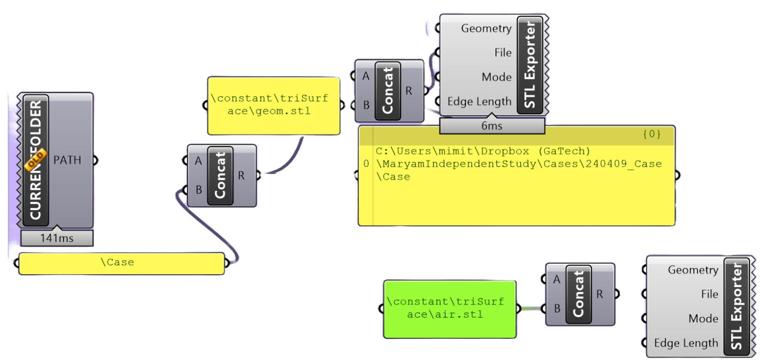


Figure 4.7: GH STL Workflow

28.this figure is not useful

GH Point in Mesh

The workflow in Figure 4.8 automatically defines the point coordinates of each region in the mesh. Each region must have a point in Rhino and the points each are connected to the corresponding material component. The *locationInMesh* parameter identifies a specific point within the computational domain, allowing the *snappyHexMesh* tool to identify and locate all cells that are connected to that point. This helps ensure that only certain regions/zones are kept during the mesh generation process, usually to separate different volumes or regions of interest.

In our case, one point is not enough to capture all the 10 regions, so, using the *locationsInMesh* is essential. This allowed us to specify a list of points that represent various locations within the mesh. Each point in this list creates a separate cell zone located in the constant/ConcreteTop/polyMesh/cellZones file, allowing more complex meshing and region differentiation. This flexibility is useful for our case and other complex geometries within a single mesh.

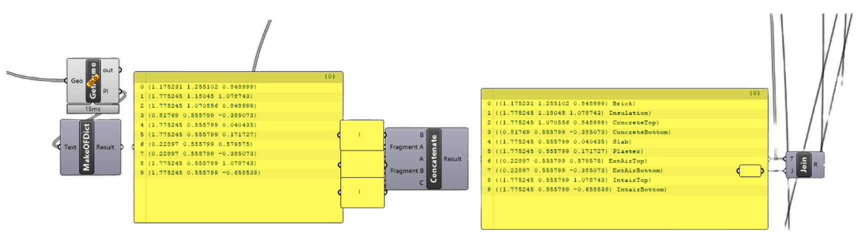


Figure 4.8: GH Point in Mesh Workflow

Block Mesh Boundary

Figure 4.9 includes a component similar to the materials component but is responsible for automatically writing the boundary conditions of minx,maxx,miny,maxy,minz, and maxz.

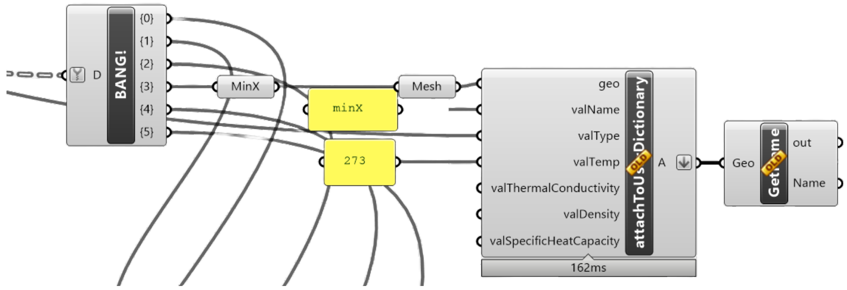


Figure 4.9: GH Boundary Conditions Component

Surface Combinations

The surface combinations workflow shown in Figure 4.10 writes the zones interfaces to be used in each zone text file in constant, system, and 0.

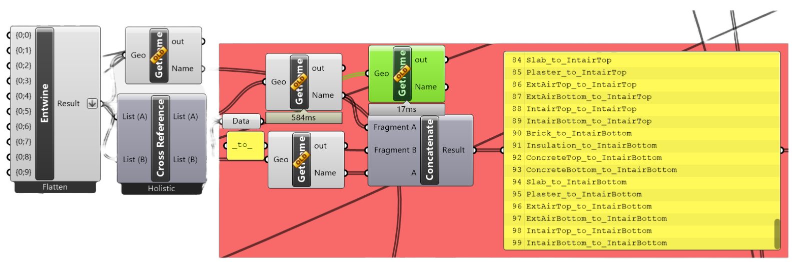


Figure 4.10: GH Surface Combinations Workflow

29.I would not show screenshots of your script if you keep it so disorganized.

Write P File Example

The sample in Figure 4.11 writes all of the P files needed to run the simulation for each zone. All of the other required text files such as T and U are written using the same workflow where the component includes a script that writes the text file in the OF required format. Also, all of the needed boundary conditions and material specifications are taken from the material dictionary component in Figure 4.6.

30.this figure is not useful

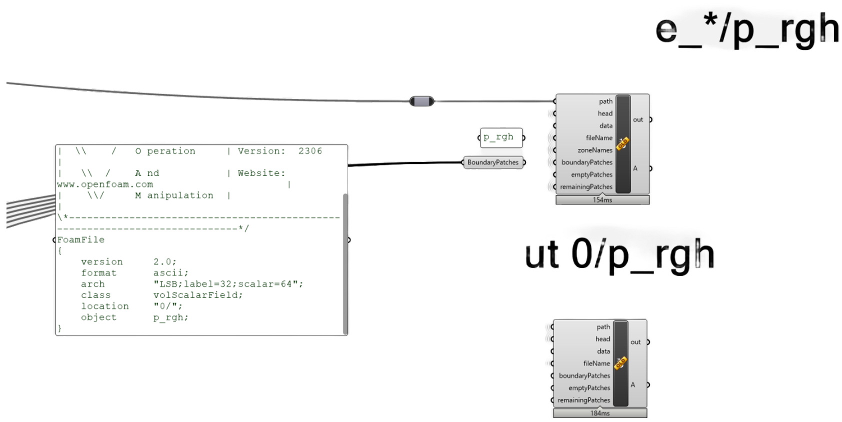


Figure 4.11: GH Write P File Workflow

4.3.6 Running The Case

After discussing the selected solver capabilities to run the numerical simulation. The goal is to solve heat transfer in solid and liquid regions between air regions and the selected geometry. Using the software *QuickField* to solve for 3D heat transfer showed some limitations where it does not consider the air regions. However, our approach using OpenFOAM includes the integration between air and solids to produce more accurate results.

Figure 4.13 illustrates the executables that need to be executed in order for a multi-region case. The *blockMesh* utility generates parametric meshes incorporating grading and curved edges. The meshes are created based on the specifications outlined in a dictionary file named *blockMeshDict* within the case directory. *surfaceFeatureExtract* extracts the surface features and then records them in a file. *snappyHexMesh* is a utility in OpenFOAM that generates 3D meshes with hexahedra from STL surfaces, iteratively refining while maintaining surface conformity. *splitMeshRegions* divides the mesh into separate regions. Each region consists of cells reachable without crossing boundary faces, including cell zones. *chtMultiRegionFoam* is a solver for both steady and transient fluid flow, with solid heat conduction and conjugate heat transfer. The equations for each system variable are solved, and the solutions from the preceding equations are inserted into the subsequent

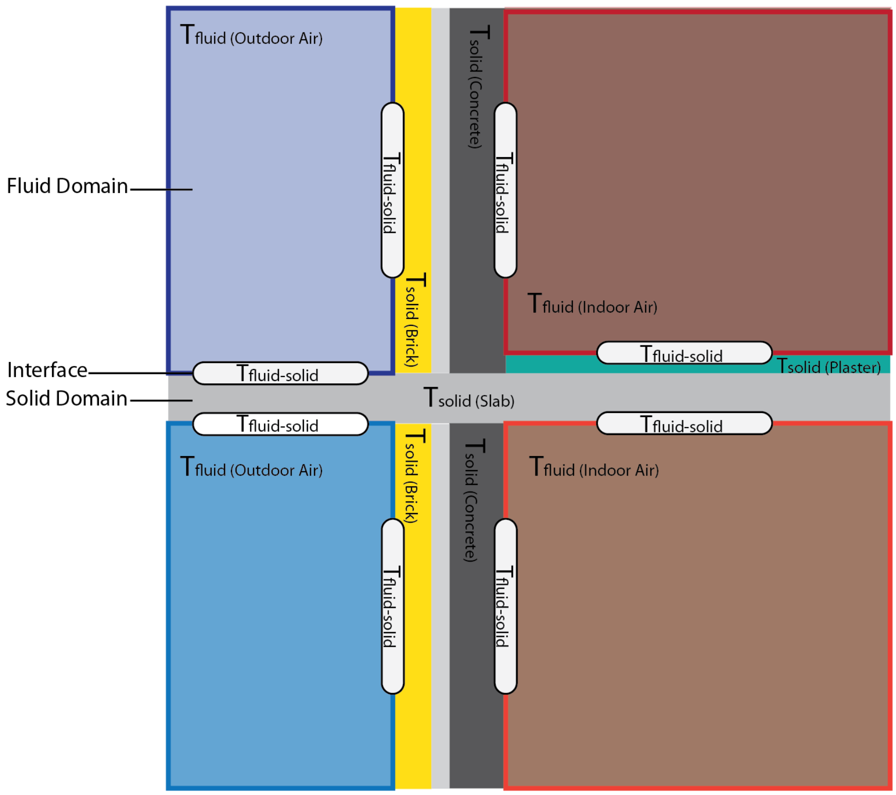


Figure 4.12: Vertical section of the validation case shows convection and conduction.

ones. For instance, fluid-solid coupling solves fluid equations first, using the previous iteration's solid temperature to set fluid temperature boundary conditions. Then we solved the fluid equations using the same method. The simulation using this process continued until convergence at time step 400.

4.4 Post-processing

The post-processing section consists of two methods to visualize the data. The first method is using *Paraview* to visualize the outcome and process the results. The second method is *OpenFOAM*'s post-process command, first, select the probing locations using GH automation components to automatically write the post-processing file and measure the data, then plot the data to compare the validation case temperatures with our results.

(WRITE HOW YOU DID POST PROCESSING AND ADD IMAGES)

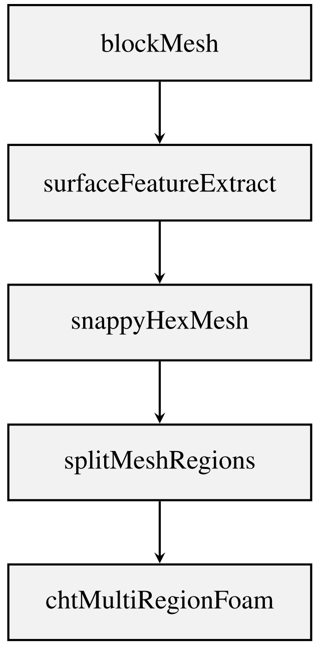


Figure 4.13: Order of executables for multiregion case setup with OpenFOAM v2306.

4.5 Results

This section discusses the results of key experimental results of our ongoing research. fig. 4.14 **(a)** visualizes the simulation temperature output, including the air (fluid) regions. Whereas fig. 4.14 **(b)** shows the geometry excluding the air, which represents the temperature distributions. The resulting temperatures are shown in the plot in fig. 4.14. The plot includes the *QuickField* outputs and our OpenFOAM simulation along the z-axis from $z = 1.2$ to $z = 2.2$ (the red line shown in fig. 4.14 **(a)**) and represents non-linear progression as a result of the different specific heat capacities and thermal conductivity of the materials. The peak in z=1.8 refers to the temperature of the slab, where it can be modified by readjusting the slab model and re-simulate.

31.Remove (c) from this figure and put it in separate figure.

4.6 Discussion

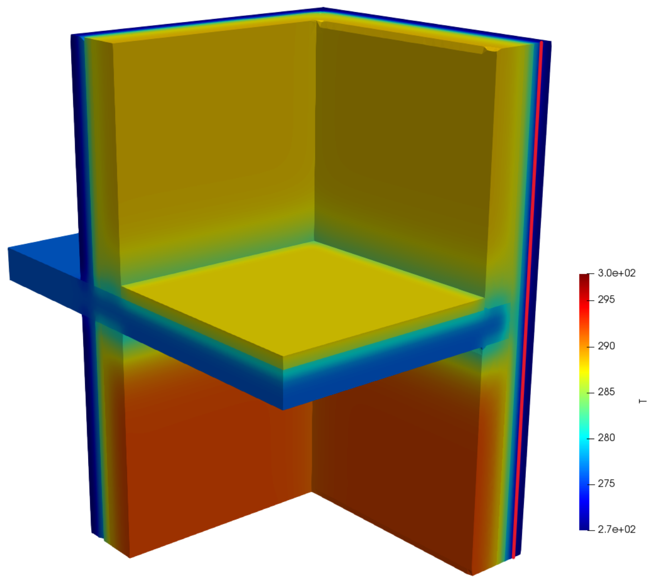
The results illustrate the capabilities of using Rhinoceros and Grasshopper to simplify the pre-processing and post-processing steps to run the 3D heat transfer simulation in Open-FOAM. The simulation focuses on convective and conductive heat transfer for a 3D heat transfer problem. **32.describe case study. It is a corner section of a building with a balcony** In some cases running the solver results in a diving by zero error which is SIFPE, but, the Grasshopper script needs to be recomputed due to opening a new Rhino file or making changes in the text files in the case path. **33.don't just say that! describe where the error exactly comes from!!! just saying some cases, doesn't help anyone.**

The experimental design section can be easily validated by setting up a transient heat transfer case using the same *chtMultiRegionFoam*. However, due to time constraints, three timesteps were chosen from the experimental results to be simulated in three separate steady-state heat transfer cases. The results are shown in Table 3.1, Table 3.2 and 13. **34.start by saying what you have done, then say what more you could have done. say the CHT solver has transient capabilities and that you can provide time-dependent boundary conditions to leverage that capability.**

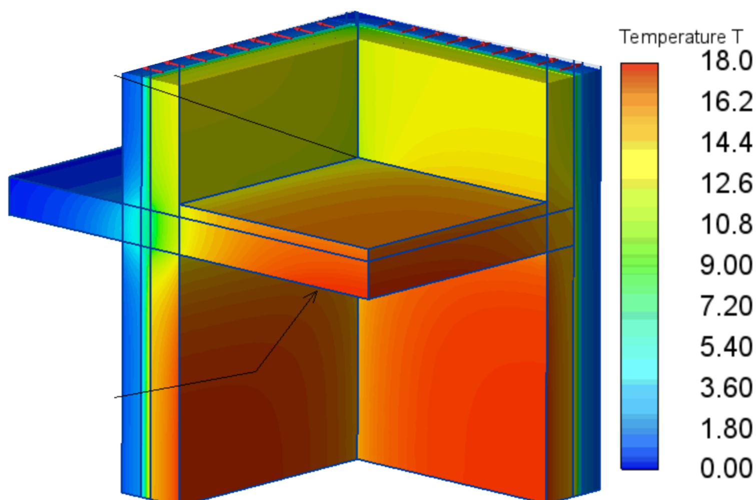
Additional potential to the simulation is enabling the users to visualize the thermal comfort in the space (temperature of fluid in the space). Also, the use of the adaptable *chtMultiRegionFoam* solver is designed to simulate complex heat transfer scenarios for various applications. It is suitable for analyzing and helping model the performance of heating and cooling systems in the HVAC industry. The automotive industry also uses the solver to better understand engine cooling. Other uses for this solver include evaluating solar heat gain and analyzing heat exchangers. **35.dont talk about automotive here, just focus on how you could use your workflow and improved versions of it for buildings**

Using *QuickField* to post-process the validation case presents several shortcomings when compared to the OF simulation shown in this paper. These limitations include the inability to customize the probing locations, limited visualization capabilities, and the absence of point density adjustment in the temperature plots. These constraints affect and limit us to compare specific temperature plots. Also, the limitations impacted the accuracy and flexibility of the 3D heat transfer simulations conducted by *QuickField*. **36.I do not understand that.**

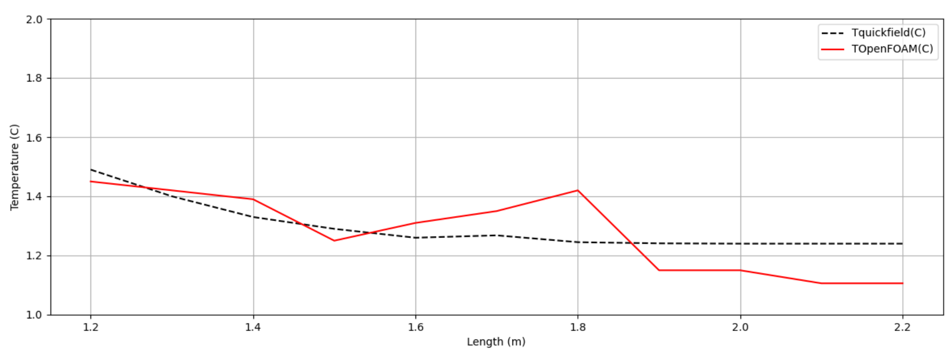
The workflow presented in the paper is an extension of the work on 2D conductive heat transfer using OF in Solving Thermal Bridging Problems for Architectural Applications with OpenFOAM by [6]. **37.sounds like you should start with this sentence. Did you look at my research seminar materials that I sent you?!** The findings highlight several issues in the field, such as cost-effective software, and disconnection between architectural modeling software with heat transfer software. Finally, the simulation is currently automated, but the goal is to provide it as a plug-in tool in *Grasshopper*.



a



b



c

Figure 4.14: Validation Plots. **(a)** **(b)** **(c)**

CHAPTER 5
CONCLUSION

This thesis presents a noticeable advancement in the field of sustainable architecture and building physics by introducing an automated integration between GH, Rhino, and OF to present thermal performance analysis in the architectural design process. **38.this sounds like Chat-GPT, just say what you have done. You built a workflow to simulate 3D heat transfer problems for architectural applications.** By utilizing OF, this research fills a critical gap in current design practices and is capable of improving energy-efficient building design and data-driven material selection. Future work in this field includes packaging the GH automation to provide a user-friendly GH plug-in. Leveraging advanced computational tools like OpenFOAM and integrating them seamlessly into existing design software offers a practical solution to a challenge and provides architects with the means to make informed decisions easier and faster. **39.please end with future work. Did you look at my research seminar material?** Decisions include, suitable material selection, thermal comfort analysis, WWR, orientation, insulation, and many more. **40.list all important ones, don't say many more.** The stated aspects all are determined using data-driven decisions by either using machine learning, optimization, or analysis software. Thus, having an integrated 3d heat transfer tool incorporated from the preliminary design phase to the schematic phase is essential to ultimately reduce the building's operational and embodied carbon emissions.

5.1 Research Outcomes

First, this research won the Kendeda Microgrant featured in [15] where the funds were used to purchase the U-value measurement kit. **41.add URL to microgrant website** The outcomes of this simulation experiment were presented at the XXX in Spring 2024. A condensed version of this thesis was has been accepted for submission in the 2024 International Building Physics conference proceedings in Toronto, Canada [16]. **42.add the micro grant support also for the 2D chapter.**

Appendices

APPENDIX A

EXPERIMENTAL EQUIPMENT

Table A.1: U-value measurement kit specifications by GreenTeg.

- **Product:** gSKIN® KIT-2615C (calibrated)
- **Article Number:** A-163479
- **GSKIN® KIT Includes (For More Details Consult The Datasheets Of The Individual Products):**
  - Sensor: gSKIN®-XO 67 7C (30mm x 30mm)
  - Logger: gSKIN® DLOG-4231
  - Double-sided mounting tape (MOUNT-1235)
- **Heat Flux:**
  - Range Min / Max [W/M²]: ±300
  - Resolution [W/M²]: < 0.22
- **Temperature Accuracy [°C]:**
  - ±0.5 (-10...+46 °C)
  - ±2.0 (-55...+125 °C)
- **Min. Sensor Sensitivity** 7.0 (sensor calibration data already loaded onto logger for simple and fast plug-and-play measurements)
- **Data Storage Capacity # Measurements]:** $<$ 2’000’000
- **Battery Lifetime [Days]:** $<$ 30 at lowest measurement frequency (2/d). Rechargeable.
- **Software:** Installation - SW sent by email / via download link
- **Logger Dimensions (mm):** 52 x 20 x 15
- **Bit Resolution [Bits]:** 12
- **Operation Modes:** Autonomous / Live
- **Operating Temperature Range Min/Max [°C]:** -40 / 100 (-25 / 65 for Logger)
- **Operating System:** Windows 2000 / XP / Vista / 7 / 8
- **Calibration Accuracy $\mp$%** 3 (Sensor calibration data already loaded onto logger for simple and fast plug-and-play measurements)
- **Calibration Temperature Range Min/Max [°C]:** -30 / 70
- **Heat Flux Sensor Cable Length [m]:** 1.5 (with connector)
- **Temperature Sensor 1 / 2 Cable Length [m]:** 5.0 / 1.0
- **Measurement Frequency:** 1/s to 1/h
- **Computer Interface:** USB

APPENDIX B
T FILE 0 EXAMPLE

```
/*---------------------------------*- C++ -*----------------------------------*\
| =========                 |                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2306                                  |
|   \\  /    A nd           | Website:  www.openfoam.com                      |
|    \\/     M anipulation  |                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    arch        "LSB;label=32;scalar=64";
    class       volScalarField;
    location    "0/";
    object      T;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

dimensions      [ 0 0 0 1 0 0 0 ];

internalField   uniform 270;


boundaryField
{
Brick_to_Brick
{
            type          compressible::turbulentTemperatureRadCoupledMixed;
            value  uniform 273;
            Tnbr T;
            K solidThermo;
            kappaMethod solidThermo;

}
```

Listing B.1: OF text file in 0/Brick/T

APPENDIX C
FILE CONSTANT EXAMPLE

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  v2306                                 |
|   \\  /    A nd           | Website:  www.openfoam.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      thermophysicalProperties;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

thermoType
{
    type            heSolidThermo;
    mixture         pureMixture;
    transport       constIso;
    thermo          hConst;
    equationOfState rhoConst;
    specie          specie;
    energy          sensibleEnthalpy;
  }

mixture
{
    specie
    {
        molWeight   12;
    }
    transport
    {
        kappa   80;
    }
    thermodynamics
    {
```

```
        Hf      1;
        Cp      1060;
    }
    equationOfState
    {
        rho     1600;
    }
}


// ************************************************************************* //
```

Listing C.1: OF text file in constant/Brick/thermophysicalproperties

APPENDIX D
FILE SYSTEM EXAMPLE

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  v2306                                 |
|   \\  /    A nd           | Website:  www.openfoam.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      fvSolution;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

solvers
{
    "(rho|rhoFinal)"
    {
        solver          PCG;
        preconditioner  DIC;
        tolerance       1e-7;
        relTol          0;
    }

    p_rgh
    {
        solver           GAMG;
        tolerance        1e-7;
        relTol           0.01;
        smoother         GaussSeidel;
    }

    p_rghFinal
    {
        $p_rgh;
        tolerance        1e-7;
        relTol           0;
```

```
    }
```
Listing D.1: OF text file in System/Brick/fvsolution

APPENDIX E
WRITE T FILE EXAMPLE

```
String directoryPath = path + @"\0\";

  if (!Directory.Exists(directoryPath))
  {
    Directory.CreateDirectory(directoryPath);
  }


  String filePath = directoryPath + @"\" + fileName;


  A = filePath;



  File.Delete(filePath);

  using(var tw = new StreamWriter(filePath)) // new StreamWriter(path,
  true) for append
  {
    tw.WriteLine(head);
    tw.WriteLine(data);




    tw.WriteLine("boundaryField");
    tw.WriteLine("{");
    foreach (Mesh  m in zoneNames)
    {
      //tw.WriteLine(str);
      //tw.WriteLine("{");
      tw.WriteLine(m.UserDictionary["name"]);
      tw.WriteLine("{");
      tw.WriteLine("  type " + m.UserDictionary["type"] + ";");
      tw.WriteLine("  value uniform " + m.UserDictionary["temp"] + ";"
 );
      tw.WriteLine("}");
      //tw.WriteLine("}");
    }
    foreach (GeometryBase geo in bMBoundaries)
    {
      tw.WriteLine(geo.UserDictionary["name"]);
      tw.WriteLine("{");
      tw.WriteLine("  type " + geo.UserDictionary["type"] + ";");
      tw.WriteLine("  value uniform " + geo.UserDictionary["temp"] + "
 ;");
```

```
        tw.WriteLine("}");
      }

      foreach (GeometryBase geo in actualBoundaries)
      {
        tw.WriteLine(geo.UserDictionary["name"]);
        tw.WriteLine("{");
        tw.WriteLine("  type " + geo.UserDictionary["type"] + ";");
        tw.WriteLine("  value uniform " + geo.UserDictionary["temp"] + "
;");
        tw.WriteLine("}");
        tw.WriteLine(geo.UserDictionary["name"].ToString().Split(new
string[] { "_to_" }, StringSplitOptions.None)[1] + "_to_" + geo.
UserDictionary["name"].ToString().Split(new string[] { "_to_" },
StringSplitOptions.None)[0]);
        tw.WriteLine("{");
        tw.WriteLine("  type " + geo.UserDictionary["type"] + ";");
        tw.WriteLine("  value uniform " + geo.UserDictionary["temp"] + "
;");
        tw.WriteLine("}");
      }

      tw.WriteLine("}");



      tw.Close();









    }
```

Listing E.1: Code in GH to write OF text file in 0/Brick/T

REFERENCES

[1] ISO, “ISO 10211:2007 thermal bridges in building construction. test case A.3 validation,” International Organization for Standardization, ISO Standard 65710, 2017, Available online.

[2] D. Rüdisser, *HTflux Simulation Software*, Jan. 2018.

[3] H. Glaser, “Graphisches Verfahren zur Untersuchung von Diffusionsvorgängen,” *Kältetechnik*, vol. 11, no. 10, pp. 345–349, 1959.

[4] S. Yang, T. Pilet, and J. Ordonez, “Volume element model for 3D dynamic building thermal modeling and simulation,” *Energy*, vol. 148, pp. 642–661, 2018.

[5] V. Gerlich, K. Sulovská, and M. Zálešák, “COMSOL Multiphysics validation as simulation software for heat transfer calculation in buildings: Building simulation software validation,” *Measurement*, vol. 46, no. 6, pp. 2003–2012, 2013.

[6] P. Kastner and T. Dogan, “Solving Thermal Bridging Problems for Architectural Applications with OpenFOAM,” in *SimAUD 2020 Proceedings*, 2020, pp. 405–412.

[7] W. Zhong, T. Zhang, and T. Tamura, “Cfd simulation of convective heat transfer on vernacular sustainable architecture: Validation and application of methodology,” *Sustainability*, vol. 11, no. 15, 2019.

[8] T. L. Bergman, A. S. Lavine, F. P. Incropera, and D. P. Dewitt, *Fundamentals of heat and mass transfer*, 2011.

[9] Z. Zhai, “Energy Efficient Buildings: Fundamentals of Building Science and Thermal Systems,” in *Wiley*, 2023, chapter 1.

[10] *GreenTeg. (n.d.). U-value - GreenTeg*.

[11] A. Singhal, *Tutorial to set up a case for chtMultiRegionFoam in OpenFOAM 2.0.0*, Aug. 2014.

[12] OpenFOAM, *chtMultiRegionFoam Solver Documentation*, 2018.

[13] A. D. L. Vollaro, G. Galli, and A. Vallati, “Cfd analysis of convective heat transfer coefficient on external surfaces of buildings,” *Sustainability*, vol. 7, no. 7, pp. 9088–9099, 2015.

[14] OpenFOAM, *ChtMultiRegionFoam*, 2019.

[15] Keneda, *Micrograntt*, 2023.

[16] ibpc, *ibpc*, 2023.

VITA

Maryam Almaian...