Journal of Advanced Research in Fluid Mechanics and Thermal Sciences 111, Issue 1 (2023) 135-155

SEMARAK ILMU PUBLISHING

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage:
https://semarakilmu.com.my/journals/index.php/fluid_mechanics_thermal_sciences/index
ISSN: 2289-7879

Modelling 2-D Heat Transfer Problem for Different Material Combination

Su Hian Ho[1,*], Yit Yan Koh[1,2], Chong Lye Lim[3,♣], Lip Kean Moey[4]

1 School of Engineering, Faculty of Engineering and Built Environment, University of Newcastle, Australia
2 Newcastle Australia Institute of Higher Education, Singapore
3 School of Engineering and Technology, PSB Academy, Singapore
4 Centre for Modelling and Simulation, Faculty of Engineering, Built Environment & Information Technology, SEGi University, 47810, Selangor, Malaysia

ARTICLE INFO

Article history:
Received 13 August 2023
Received in revised form 24 October 2023
Accepted 2 November 2023
Available online 15 November 2023

Keywords:
Numerical method; programming; modelling heat transfer; optimum grid size

ABSTRACT

The analysis of heat transfer problems can be highly complex due to factors such as temperature, position, and time. Most heat transfers are typically two-dimensional as conduction is often negligible in the third dimension. Two-dimensional heat conduction problems can be solved analytically or numerically. In steady-state conditions, the Laplace equation can be applied to solve two-dimensional heat conduction problems analytically, in which the separation of variables method is used to solve the Laplace equation under fixed boundary conditions to determine the temperature at a specific point. The Laplace equation plays a significant role in the solution of heat transfer problems, as it demonstrates the behavior of linear and non-linear equations in the computational fluid dynamics domain. Despite their inability to provide exact results at any point, numerical methods are superior to analytical methods when handling complex geometries with various boundary conditions. This project involves the development of a computational code using MATLAB to solve two-dimensional steady-state heat conduction problems using Gauss-Seidel iterations. Comparing analytical solutions from Excel with numerical solutions from MATLAB and ANSYS, specifically the developed MATLAB code, revealed an accuracy level of 99.902% for the Laplace equation. An analysis of the produced code from MATLAB found that it could solve two-dimensional steady-state heat conduction across different combinations of materials while allowing users to specify initial and boundary conditions to produce a contour plot similar to ANSYS.

1. Introduction

Recent advances in modelling and simulation techniques, as well as the use of high-speed computers, have made it less challenging to conduct thermal and heat transfer analyses. More models are being created, tested, and used, which simplifies the calculation process and allows for both immediate results and the prediction of future trends and other auxiliary data. For conventional

* *Corresponding author.*
E-mail address: suhian.ho@uon.edu.au
♣ *Corresponding author.*
E-mail address: chonglye.lim@psb-academy.edu.sg

https://doi.org/10.37934/arfmts.111.1.135155

and advanced processes, simulation methods and simulation software like MATLAB, ANSYS, and SimScale may be used to tackle heat transfer-related issues [1].

Due to the limitations of mathematical techniques, analytical solutions to thermal conduction differential equations are frequently difficult. Numerical techniques for resolving the heat conduction problem have long been a study interest due to the development of computer technology. In the past few decades, heat conduction problems have been effectively resolved utilizing traditional numerical techniques, including the boundary element, finite volume, and finite difference approaches. The complexity of engineering techniques is substantially increased by the fact that, despite the methods' shown correctness, solving discrete equations for heat conduction problems still necessitates a sizable amount of integral computations. A concentrated effort has thus been undertaken to research and create novel numerical solution techniques [2].

Li *et al.,* [2] suggested a hidden temperature approach to handle data-driven computational issues involving one-dimensional heat conduction in a recent study. During the iterative process, this novel model, which uses an artificial neural network, creates a correspondence link between the node temperature values, yielding a "Data to Data" answer. Li *et al.,* [2] compared the hidden temperature method with conventional numerical methods and concluded the hidden temperature method provides highly accurate in both steady-state and transient conditions.

Matt and Cruz [3] presented a study on two-phase composite heat transfer with a finite element computational method. This study also emphasized the need for further geometric and physical model development to produce more accurate computational findings, particularly in the context of random (disordered) composites' three-dimensional heat conduction. The gap in benchmark results for effective conductivity in random composites has been emphasized by Matt and Cruz [3].

Building upon these prior studies, the present research aims to contribute to the field of numerical heat transfer analysis by providing an alternative method to solving two-dimensional heat transfer. This study aims to develop a computational code that estimates an optimum grid size for steady-state heat conduction in two dimensions using suitable iterative methods. There are typically no significant differences in the results obtained after a certain grid size, therefore, fewer iterations can be undertaken to save time. This code will give researchers and engineers an alternative numerical approach for efficient and accurate heat transfer analysis.

By addressing the gap in benchmark results for effective conductivity in random composites and offering a practical computational tool, this study aims to advance the understanding and application of numerical methods in heat transfer analysis.

In summary, recent advancements in modelling and simulation techniques have paved the way for more efficient thermal and heat transfer analyses. Numerical methods have gained prominence due to the limitations of analytical techniques. The hidden temperature method proposed by Li *et al.,* [2] has demonstrated high accuracy in one-dimensional heat conduction problems, while the work of Matt and Cruz [3] highlighted the gap in benchmark results for random composites' effective conductivity. Following these studies, the current work seeks to incorporate programming as a method of solving heat transfer problems while analyzing two-dimensional heat transfer across a variety of material combinations, and contribute to the field of numerical heat transfer analysis.

2. Methodology
2.1 Methodology Flowchart

The methodology, as shown in Figure 1, begins with the development of MATLAB code for simulating steady-state temperature distribution. The first step involves specifying the grid geometry and boundary conditions for the heat conduction problem. To evaluate the accuracy of numerical solutions, an analytical solution is first performed using Microsoft Excel as a reference. Simultaneously, a numerical solution is conducted in ANSYS under the same conditions. The analytical and numerical solutions are then compared to assess the agreement and identify any disparities. If the numerical solution proves to be inaccurate, refinements are made to the ANSYS model, and if it is accurate, the methodology proceeds to the development of the MATLAB code. Once the ANSYS model has been verified as accurate, MATLAB simulations are conducted under the same conditions as the Excel and ANSYS simulations. The temperature distribution results obtained from the MATLAB simulation are carefully examined to assess their accuracy and consistency.

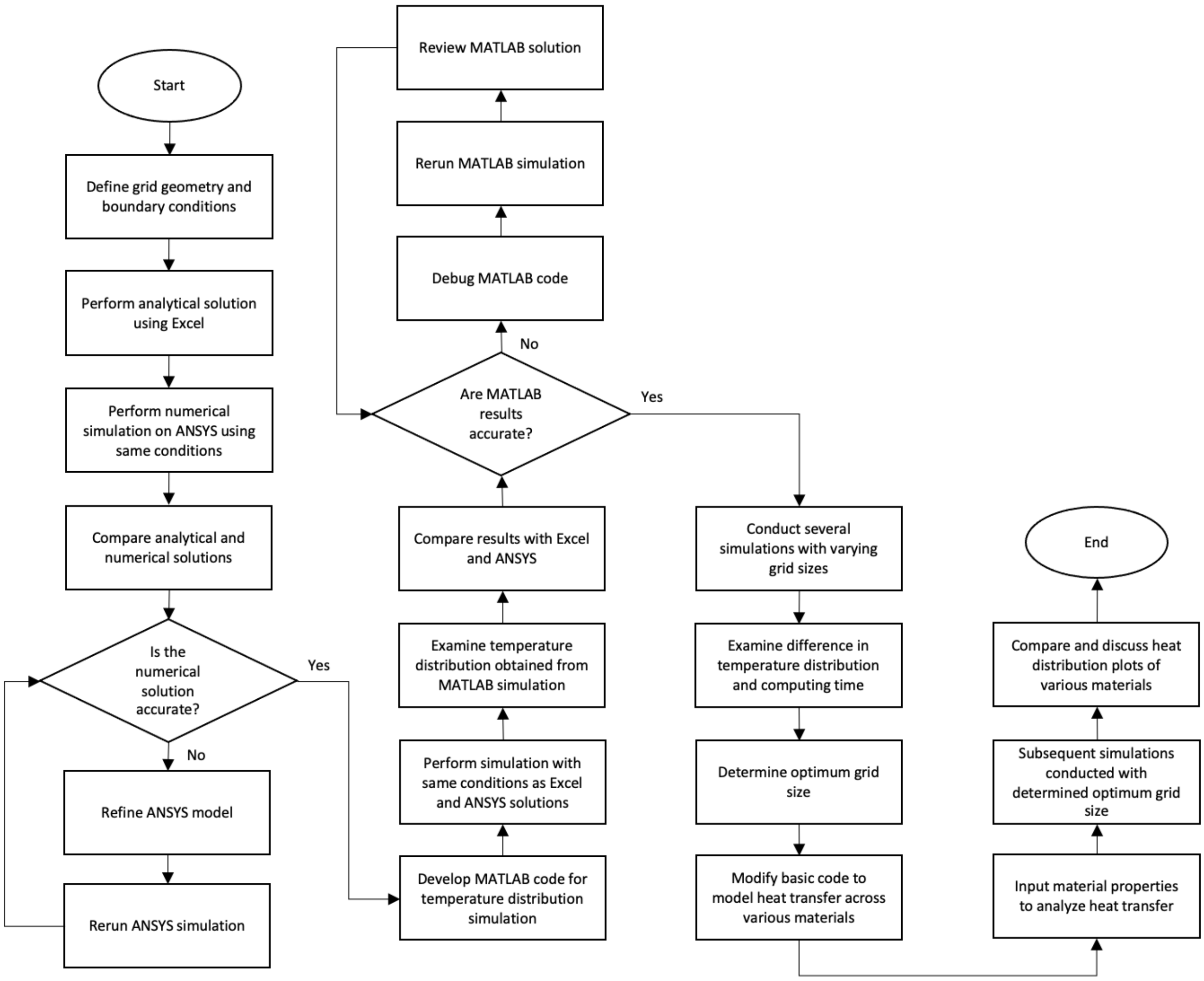

Fig. 1. Methodology flowchart

A comparison is made between the MATLAB results and those from Excel and ANSYS to determine whether they are in agreement and if any discrepancies exist. A debugging process and rerun of the simulation may be necessary if the MATLAB results are not accurate, and if they are, multiple simulations with varying grid sizes may be conducted. Analysis of the results of varying grid sizes is used to determine the optimum grid size by identifying cases with minimal temperature distribution differences and reduced computational time, thereby enabling efficient computations. The MATLAB code is enhanced to model heat transfer across materials, and material properties are input for analysis. The heat distribution plots of various materials are compared and discussed to gain a deeper insight of their behaviour and characteristics. The methodology concludes after simulating temperature distribution, optimizing grid size, and analyzing heat transfer across various materials have been achieved.

2.2 Finite Difference Method

Heat transfer problems are most commonly solved using the Laplace equation in the study of heat conduction. The Laplace equation generally describes equilibrium situations or those that do not depend explicitly on time, which essentially means steady-state. There are both analytical and numerical approaches to solving the Laplace equation. In the analytical approach, there are methods like the integral heat balance method and spreadsheet programs like Microsoft Excel to calculate temperature distributions. In the numerical approach, solutions are found using numerical methods such as the finite difference, finite element, or boundary element methods [4]. Due to its simplicity in implementation, finite difference methods have been widely used in solving heat conduction problems. This project utilizes the finite difference method to discretize the Laplace equation.

Finite difference methods begin by discretizing space and time coordinates to create a mesh of nodes. A set of linear algebraic equations (termed nodal equations) can then be derived from the energy balance applied to the volume elements surrounding the nodes. These equations, containing as many unknown as nodes in the mesh, are solved through matrix factorization or iterative methods. In order to obtain the finite difference solutions, computer programs are used since the accuracy of the approximation increases with the number of nodes [5]. Based on this, the accuracy of the approximation is inextricably tied to the grid resolution. As grid resolution increases, smaller grid squares are created, resulting in more accurate results at the cost of additional computational resources.

Figure 2 illustrates a section of a body subdivided into several small volumes using equal divisions in the *x*- and *y*- directions. Nodal points (nodes) of these volumes are denoted with *(m+1, n)*, *(m, n)*, *(m-1, n)*, etc., where *m* and *n* represent *x* and *y* increments respectively. At the nodal points of each volume, it is assumed that the thermal properties are concentrated. A finite difference approach is used to approximate the Laplace equation's differentials to obtain temperature at the nodal points [5]. Nodal equations should be written for each node, with temperature as the unknown. Once the equations are derived, the temperature at each node is obtained by solving them [6].

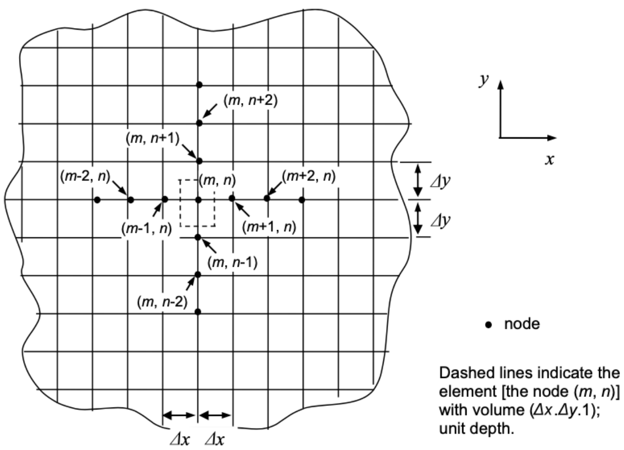

Fig. 2. Defined nomenclature for two-dimensional numerical analysis [5]

2.3 Numerical Modelling and Simulation using ANSYS

This section delves into numerical modelling and simulation using ANSYS, providing valuable insights into temperature distribution and comparing the nodal temperature analytical solution with that of numerical modelling. Simulations of temperature distribution were conducted using the ANSYS simulation software to compare the calculated nodal temperature analytical solution with the simulated numerical temperature distribution. Based on the simulation, a block with dimensions of 1m-by-1m was used, and since the three edges are of the same temperature, the three edges are assumed to be insulated. Consequently, there is no conduction or convection on those sides, except for the top edge where air is present. The block dimensions were determined based on the geometry of the grid used to calculate the nodal temperature in Excel. The parameters used in both the ANSYS simulation and Excel calculation were kept the same as much as possible in order to ensure a fair comparison of results. The specifications for the simulation are provided in Table 1.

Table 1
ANSYS simulation specifications

Property	Value	Units
Left Edge Temperature	500	K
Right Edge Temperature	500	K
Bottom Edge Temperature	500	K
Top Edge Temperature	300	K
Ambient Temperature	300	K
Top Edge Heat Transfer Coefficient	40	$W/m^2.K$

To verify, validate, and benchmark the accuracy of numerical solutions like MATLAB in solving heat conduction problems, the obtained results from MATLAB were compared to ANSYS results and Excel analytical solutions. As there are no experimental results in the modelling of heat transfer across materials, it is necessary to validate ANSYS numerical data to demonstrate that the MATLAB solver is able to predict outcomes accurately and reliably by comparing them with established correlations in the literature [7]. Additionally, by comparing both analytical and numerical solutions, it can be determined if numerical methods are accurate, which, in turn, may be used to determine the computational cost in terms of grid size and time. While the numerical method does not provide exact solutions to two-dimensional heat conduction problems, it is more applicable and has sufficient accuracy to be used in solving them, as the analytical method is only suitable for solving problems with simple boundary conditions [8]. The implementation of this method would contribute to the project's objective of determining the optimum grid size by saving time on executing the code.

2.4 Analytical Calculation using Excel

In contrast to an analytical solution, which allows for temperature distribution at any point of interest in a medium, a numerical solution enables the determination of the temperature at only discrete points. Analytical solutions based on Eq. (1) would be compared with numerical solutions on ANSYS to determine which provides a more accurate distribution of temperatures. Eq. (1) was derived from a solution to the Laplace equation for two-dimensional heat conduction under steady-state conditions by using the separation of variables method.

$$\theta(x,y)=\frac{2}{\pi}\sum_{n=1}^{\infty}\frac{(-1)^{n+1}+1}{n}sin\frac{n\pi x}{L}\frac{\sinh\left(\frac{n\pi y}{L}\right)}{\sinh\left(\frac{n\pi W}{L}\right)} \tag{1}$$

Figure 3 displays the grid used for the calculation of the nodal temperature in the Excel analytical solution. In order to verify the reliability of the numerical simulation, it was necessary to perform the numerical simulation using the same boundary conditions as those shown in Figure 3, which ensured that similar results were obtained from the analytical results on Excel. Furthermore, the numerical simulation and analytical solution are used as benchmarks to assess the functionality of the MATLAB-developed code.

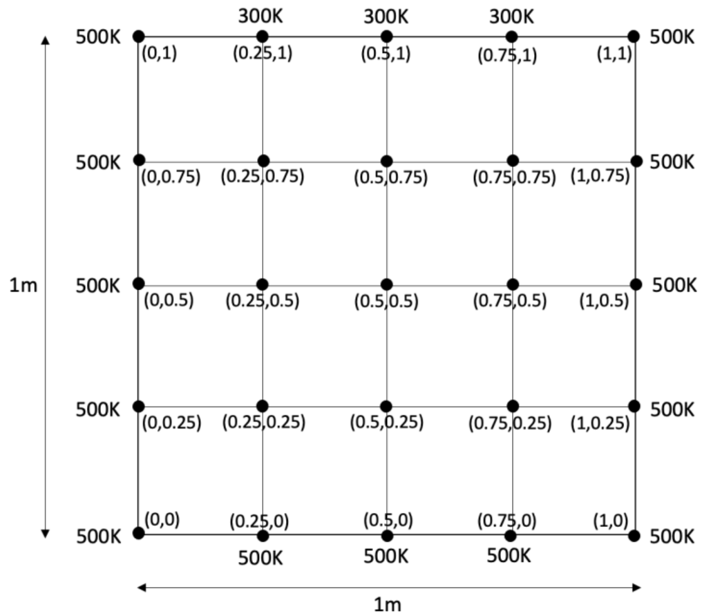

Fig. 3. Grid for nodal temperature calculation in analytical solution

2.5 Numerical Computation using MATLAB

To validate the numerical solution, MATLAB was employed as a computational tool to explore an alternative numerical approach. As a result, a numerical solution comparable to that validated through ANSYS simulations could be developed and assessed. The numerical solution, once validated by comparing the analytical and numerical solutions, can thus be developed using MATLAB as an alternative numerical method. In order to validate the MATLAB code, a simulation was performed under similar conditions to the one that was conducted with ANSYS. An essential aspect of numerical studies is the validation of numerical codes to ensure their accuracy in comparison with other previous studies. Additionally, the code validation process facilitates the test of the code prior to its execution [9]. In general, it should produce the same results or results very similar to those of previous studies. Furthermore, it is crucial to gain a thorough understanding of a numerical code's capabilities and limitations in order to achieve high accuracy [10]. A fair comparison of results was ensured by using identical geometry, grid size, and boundary conditions to those used in the ANSYS simulation. The block measures 1m-by-1m. has a grid size of 5×5, and the edges of the block display identical temperatures. A point of interest will be selected after the simulation is complete in order to determine whether the temperature at that point is similar for all three methods. By verifying the accuracy of the MATLAB code, the optimum grid size can be determined, and boundary conditions and material properties may be manipulated in order to study how each factor affects heat transfer.

3. Results

3.1 Comparison of Analytical and Numerical Solutions from Excel and ANSYS

Calculations of the nodal temperature were performed using an Excel spreadsheet, which enabled a systematic analysis of the temperature distribution. Table 2 presents the calculated temperature values at specific nodes within the block. The recorded temperature values at different nodes serve as useful reference values for subsequent validation of numerical simulation tools such as ANSYS and MATLAB.

A comparison with the analytical solution is necessary to validate the numerical solution [4]. Comparing Table 2 and Figure 4, it is clear that ANSYS exhibits a distinct difference in temperature change compared to Excel. It is evident from both figures that the temperature contour in ANSYS is much smoother, provides a better view of the distribution of heat across the block, and allows a more accurate analysis of heat transfer rates. In addition, the ability of mesh refinements for the numerical solution leads to higher accuracy in the results obtained.

Table 2
Calculated Nodal Temperature Analytical Solution using Excel

y			x		
	0	0.25	0.5	0.75	1
1	500	301.18	300.84	301.18	500
0.75	500	413.59	391.89	413.59	500
0.5	500	463.59	450	463.59	500
0.25	500	486.41	480.92	486.41	500
0	500	500	500	500	500

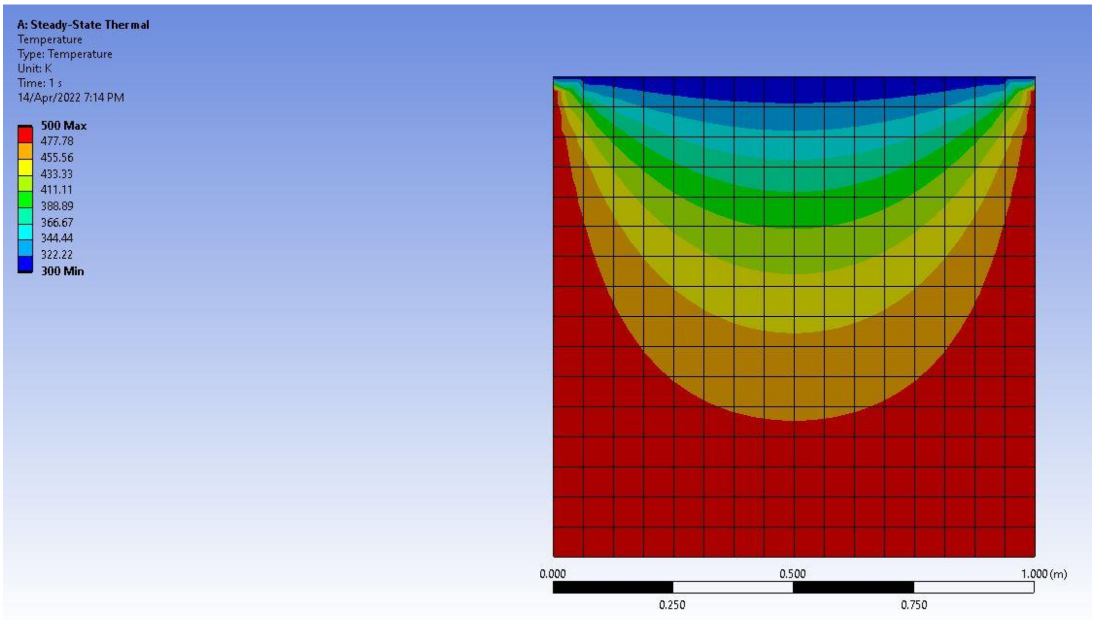

Fig. 4. ANSYS simulation with the same boundary conditions as Excel analytical solution

Mesh refinement is essential for increasing the accuracy of a numerical solution created with finite elements [11]. An iterative refinement process involves finding the solution, calculating error estimates, and refining parts of the model with high error rates. In comparison to the analytical method, the numerical method is a better solution for solving two-dimensional steady-state heat conduction problems since the accuracy of the results retrieved is higher. In a comparison of both analytical and numerical solutions, it can be concluded that both results are almost the same. If accurate results from ANSYS are desired, generating a finer mesh in the block is essential, whereby results obtained for a fine mesh can be compared with those obtained for a coarse mesh.

3.2 Validation of MATLAB Numerical Solution

In the following stage of analysis, a simulation was carried out under similar conditions done on ANSYS from the computational code developed on MATLAB. A fair comparison of results was ensured by using identical block dimensions, grid size, and boundary conditions to those used in the ANSYS simulation. The block measure 1m-by-1m, has a grid size of 5×5, and the edges of the block display identical temperatures. Based on the simulation performed under these conditions, the MATLAB code produced a plot, as shown in Figure 5, which showed similar results to those obtained from ANSYS due to the approximate similarity of the temperature distribution. Comparing the results obtained from the three methods showed that the temperatures determined at each discrete point were similar. Accordingly, this validates the accuracy of the numerical solution and the possibility of calculating temperature distributions further using this method.

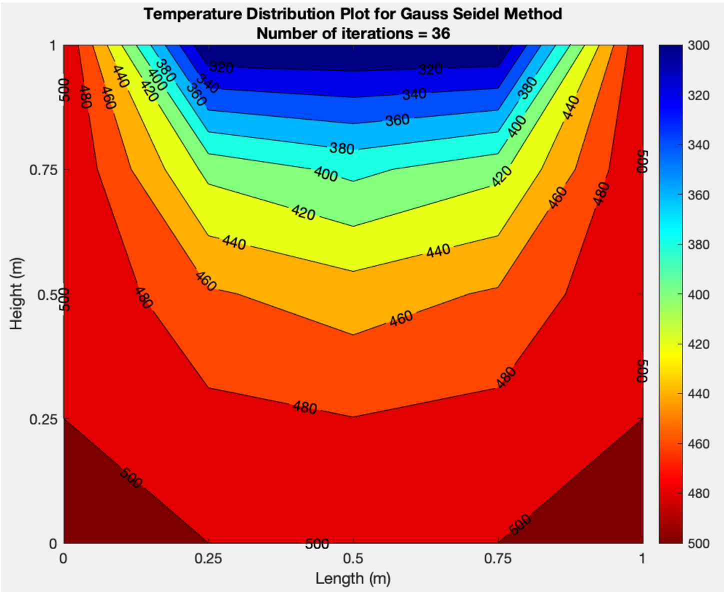

Fig. 5. MATLAB simulation with the same boundary conditions as Excel and ANSYS

3.3 Obtaining Optimum Grid Size

Verification processes have a direct impact on the validity and accuracy of the obtained outcomes. Accordingly, it is necessary to determine the optimum grid size prior to performing simulations, also known as a grid independence test in other studies [12]. Grid independence tests are generally conducted as part of the process of designing an optimal grid, because an optimal grid is required to produce accurate results. However, the approximate solution of each grid also has an impact on the accuracy of the overall simulation results. A critical factor that influences the total computational cost and the accuracy of simulation analysis results is the number of grids. Coarse grids result in significant spatial discretization errors, reducing the accuracy of the analysis results [13]. Meanwhile, too many fine grids may increase the round-off error above the truncation error, this reducing the accuracy of the results. As a result, it is imperative to determine an appropriate number of grids. In many CFD studies, grid independence tests were conducted in order to determine the optimum grid size. Based on an evaluation of several grid conditions, the grid independence test aims to find the optimal grid condition with the fewest number of grids without causing a difference in the numerical results [14].

In this project, one of the objectives was to determine the optimum grid size. In general, after a certain grid size, the results obtained will show minimal differences. It is therefore possible to save computational costs by not having to run so many iterations once an ideal grid size has been determined. Several simulations were conducted with various grid sizes in order to determine the optimum grid size. Additionally, the number of iterations was calculated at the point where the code enters the while loop, which continues until the difference between the current and previous solutions is less than the tolerance. The elapsed time for execution of the code was calculated and compared against the other grid sizes. The elapsed time can be used to determine which is the optimum grid size based on the shortest amount of time required to achieve the same results as the analytical solution [15,16].

A compilation of results obtained for various grid sizes can be found in Table 3, which shows the number of iterations required for Gauss-Seidel to reach convergence and the average time required for the code to be executed. Also shown in Figure 6 to Figure 13 are plots of error against iterations and temperature distribution contours for various grid sizes with fixed boundary temperatures. The presented results will allow an improved analysis and comparison to be made in order to determine an optimum grid size.

Table 3
Matlab numerical results of various grid sizes with fixed boundary temperature

Grid Size	Iterations	Elapsed Time (s)
5×5	36	0.007724
9×9	140	0.012809
17×17	494	0.022205
33×33	1692	0.125093
65×65	5624	0.280241
129×129	17896	3.653261
257×257	53172	39.199266
513×513	139047	566.910094

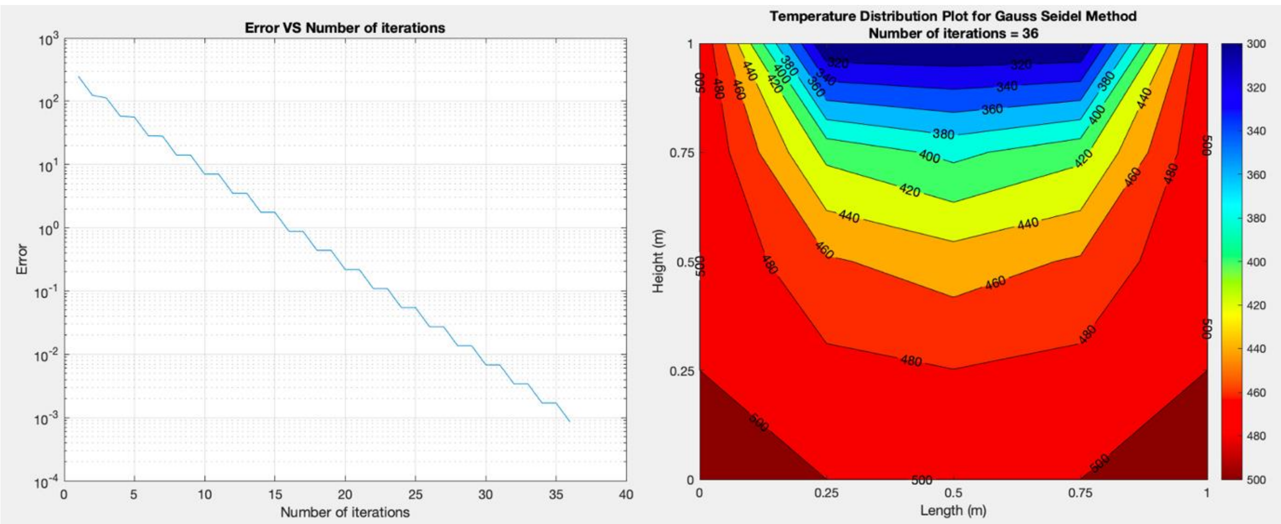

Fig. 6. Results of 5×5 Grid Size

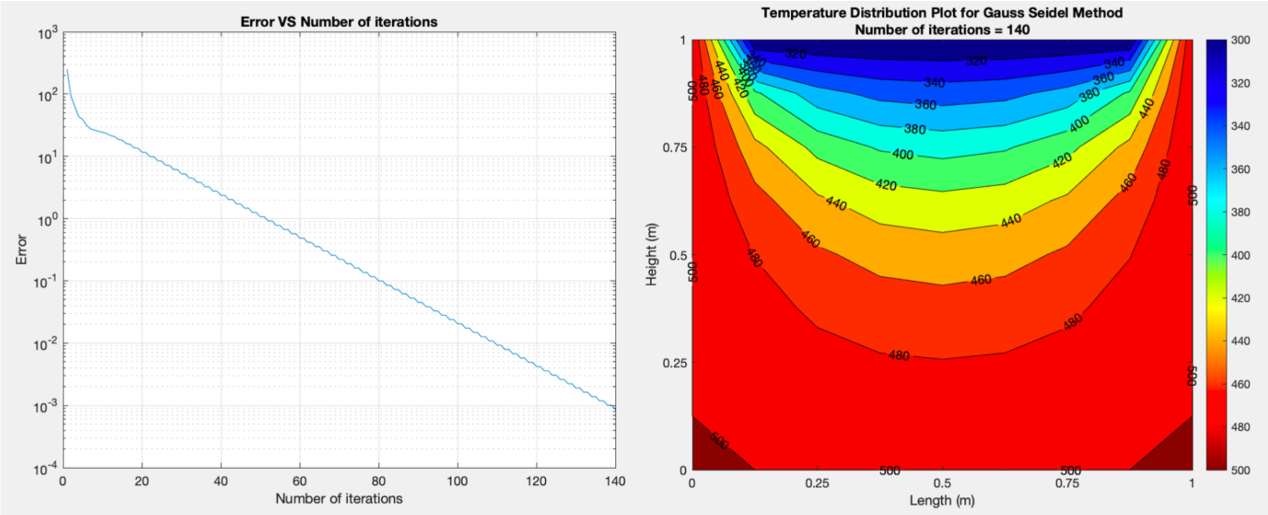

Fig. 7. Results of 9×9 Grid Size

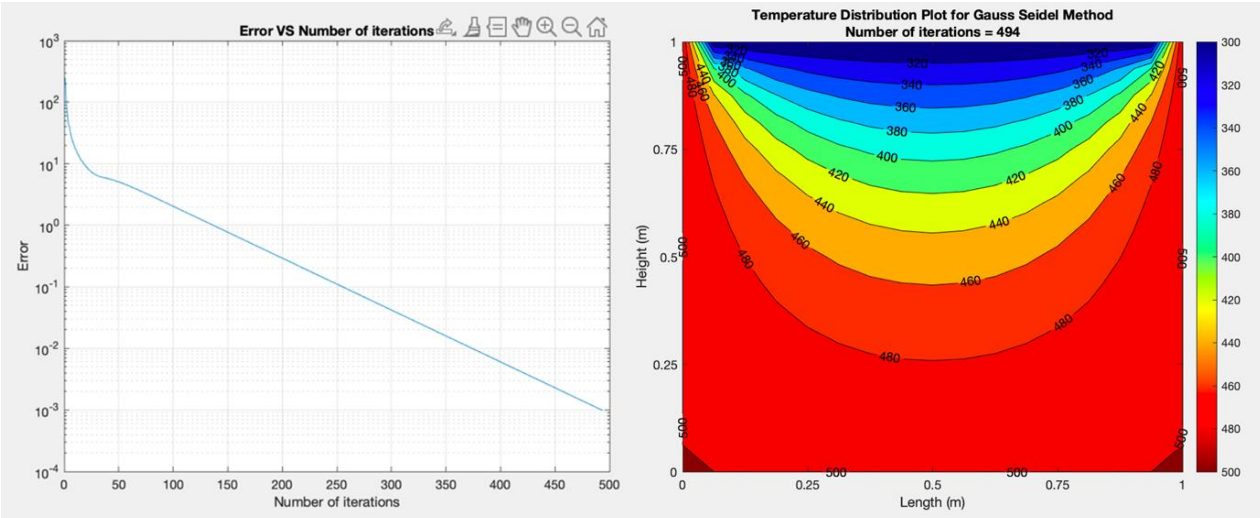

Fig. 8. Results of 17×17 Grid Size

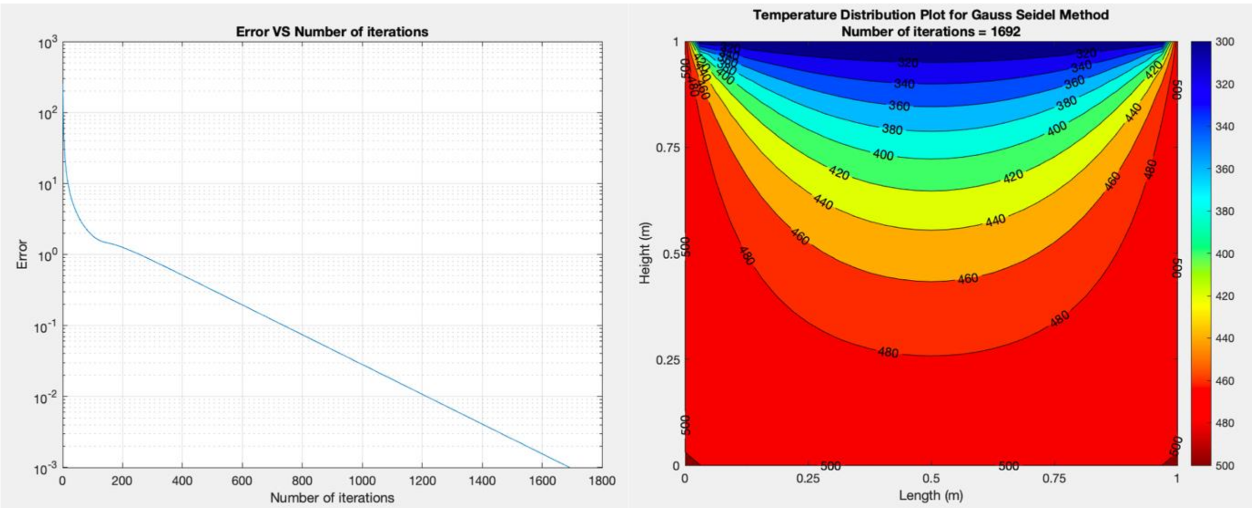

Fig. 9. Results of 33×33 Grid Size

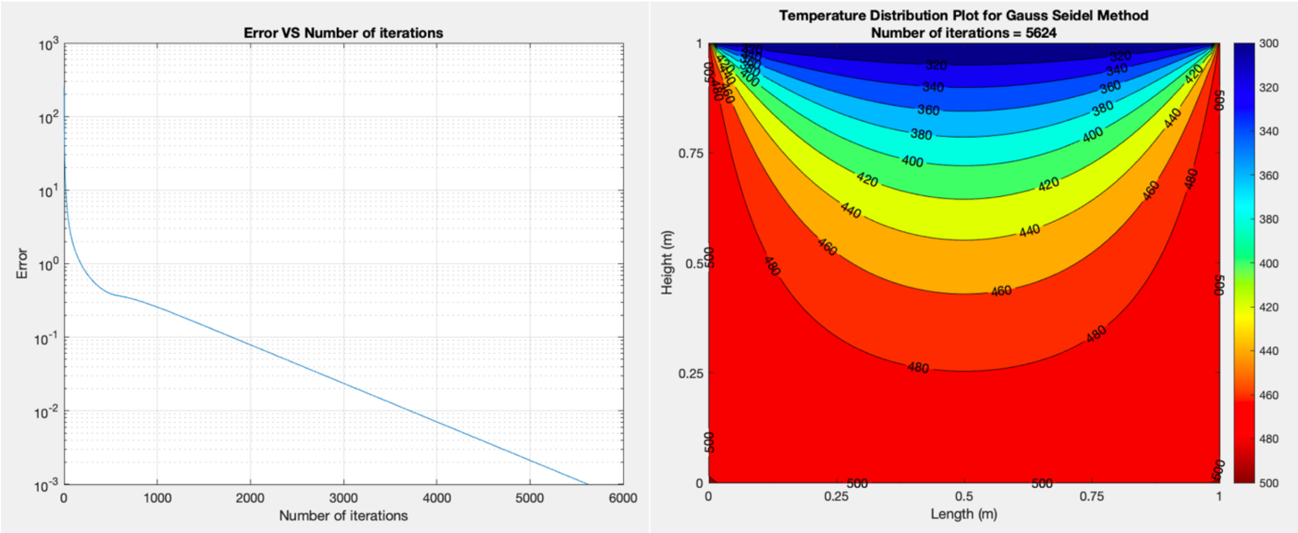

Fig. 10. Results of 65×65 Grid Size

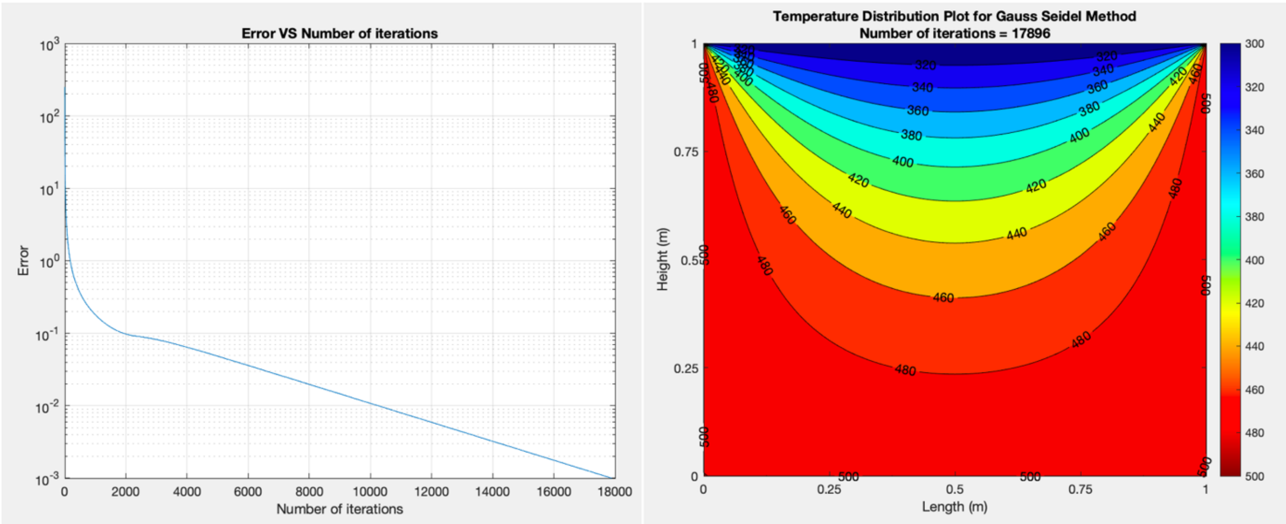

Fig. 11. Results of 129×129 Grid Size

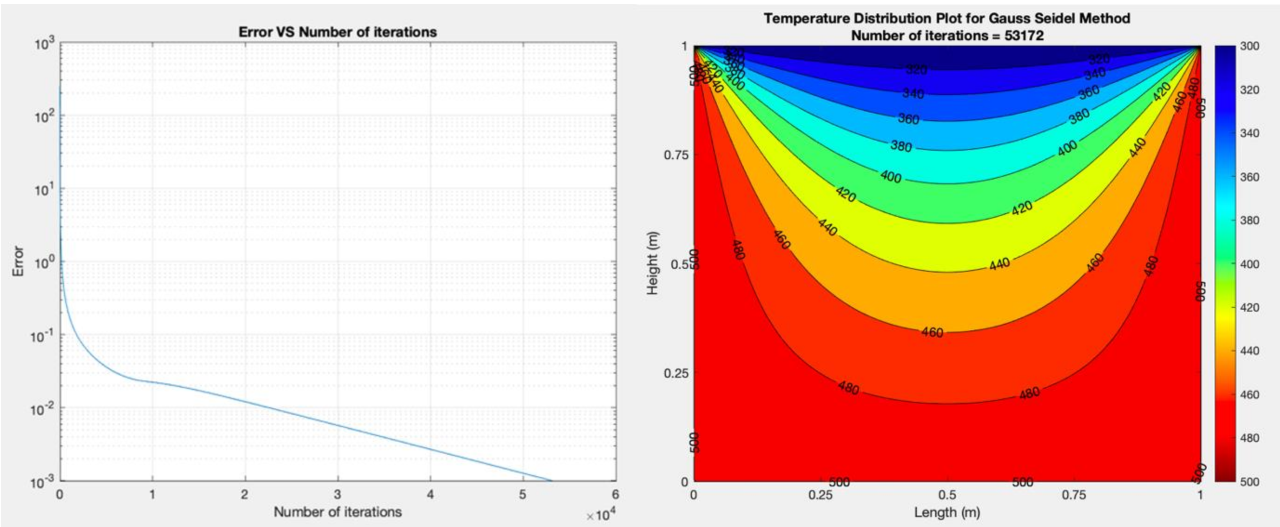

Fig. 12. Results of 257×257 Grid Size

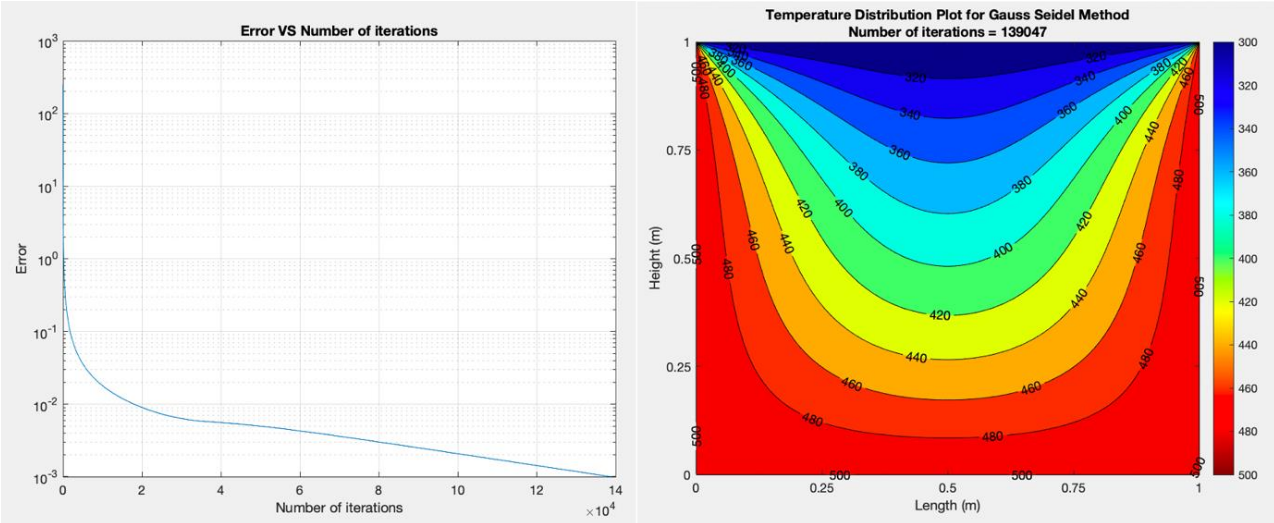

Fig. 13. Results of 513×513 Grid Size

Observing the error and temperature distribution plots, the optimum grid size appears to be 65×65. The reason for this is the fact that from observing the contour of the temperature distribution for all grid sizes after, there are no evident differences. Furthermore, it can also be observed in the graphs of increasing grid size that there is a steep decrease in error as iterations increases. However, if elapsed time were to be compared, 65×65 seems to be the optimum grid size since it takes the least amount of time to execute the code. In addition, Figure 14 supports this analysis when a reference point has been taken from the analytical solution, as shown in Table 2, to determine if the same reference point would produce the same results on the contour plot of the temperature distribution.

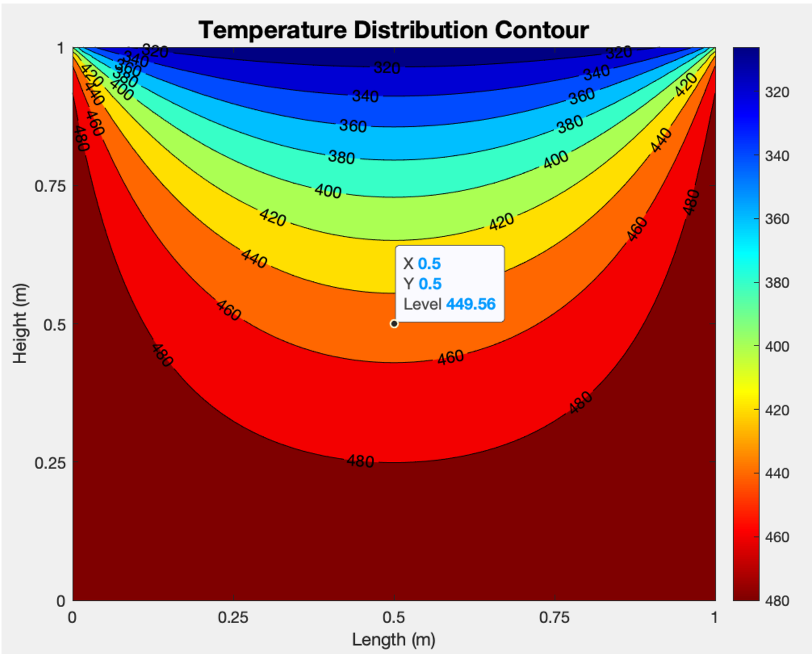

Fig. 14. Reference Point of 65×65 Grid Size

Based on the x and y coordinates equal to 0.5, the analytical solution obtained 450K as the temperature. Using the same reference point, the temperature plot for the 65×65 grid size produced a value of 449.56K, nearly the same as the analytical result. Using a 65×65 grid size can produce similar results to the analytical solution, thus saving time without running as many iterations.

3.4 Heat Transfer Across Different Materials

Simulations have been conducted on various materials to investigate the transfer of heat between them. For each simulation, the dimensions of the materials and boundary conditions have been altered to determine if there are significant differences in the contour plots of the temperature distribution. Unlike the simulations presented in the earlier sections, the simulations in this section incorporate a new thermal conductivity parameter, which measures a material's ability to conduct heat.

3.4.1 Aluminium and wood

The first test focused on aluminium and wood. The thermal conductivity values used for aluminium were 237 W/m·K and 0.17 W/m·K for wood, under the assumption that the ambient temperature is 300K [17,18]. For the first test involving these two materials, there were two simulations carried out with varying widths. In the first simulation, aluminium was simulated on the left and wood on the right, separated on a left-right axis. Aluminium and wood were simulated with widths of 0.3m and 0.7m, respectively. As part of this simulation, a fixed temperature of 500K is applied at the left boundary, convection at the right boundary, and insulation and the top and bottom boundary. Table 4 and Figure 15 illustrate the results of this simulation.

Table 4
Matlab numerical results of various grid sizes for Simulation 1

Grid Size	Iterations	Elapsed Time (s)
5×5	34	0.038627
9×9	95	0.023307
17×17	297	0.030171
33×33	961	0.106162
65×65	3141	0.445498

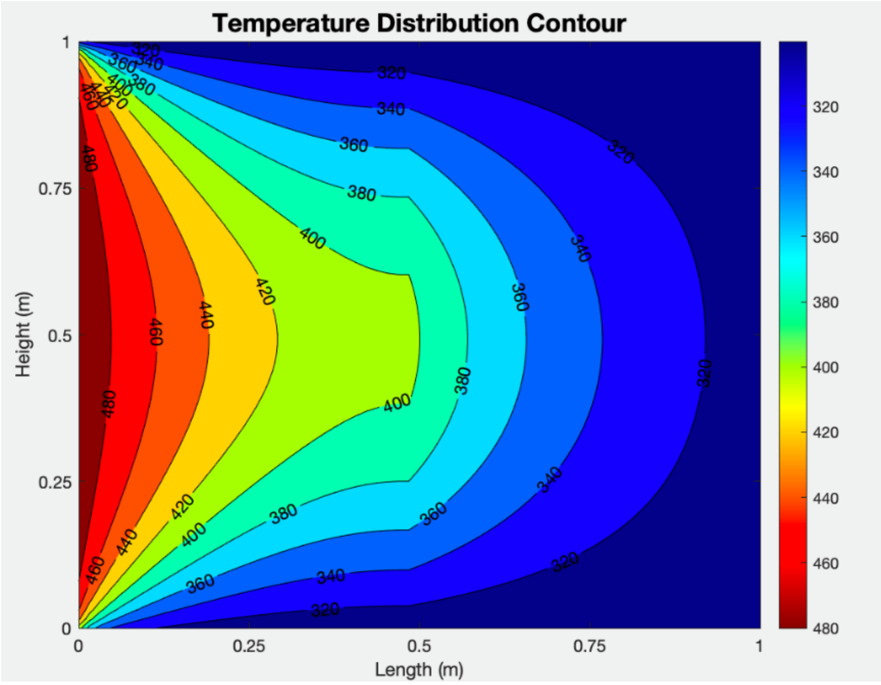

Fig. 15. 65 × 65 Grid for Aluminium (Left) and Wood (Right) – Fixed Temperature (Left), Convection (Right), Insulation (Top and Bottom)

In the second simulation, wood was simulated on the left and aluminium on the right, separated on a left-right axis. Wood and aluminium were simulated with widths of 0.3m and 0.7m, respectively. In this simulation, the same boundary conditions are applied as in Simulation 1. Table 5 and Figure 16 illustrate the results of this simulation.

Table 5
Matlab numerical results of various grid sizes for Simulation 2

Grid Size	Iterations	Elapsed Time (s)
5×5	70	0.027982
9×9	173	0.036485
17×17	481	0.048386
33×33	1384	0.213352
65×65	3983	0.744630

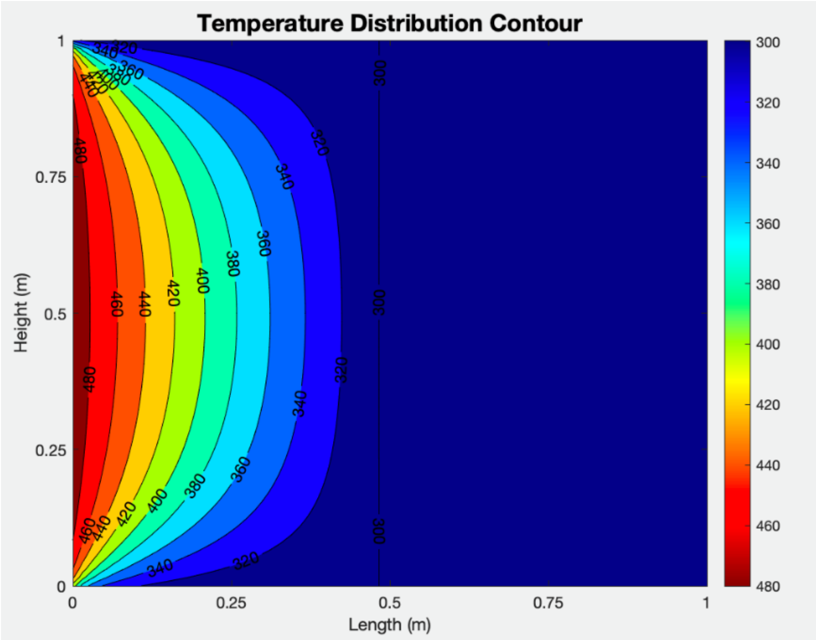

Fig. 16. 65×65 Grid for Wood (Left) and Aluminium (Right) – Fixed Temperature (Left), Convection (Right), Insulation (Top and Bottom)

Despite the same boundary conditions and material used in both simulations, both plots produced significant differences in temperature distributions. As the width of the materials was kept constant, this indicates that material placement is crucial in analyzing heat transfer across a variety of materials with different thermal properties. Furthermore, based on the analysis of the two plots, it appears that conduction is likely to dominate heat transfer between aluminium and wood in contact. A further conclusion that can be drawn is that aluminium is a much better conductor of heat than wood, owing to its higher thermal conductivity. Consequently, aluminium is able to transfer heat much more effectively than wood. It is natural for heat to flow from the object with the higher thermal conductivity to the object with the lower thermal conductivity when two objects made of different materials are in contact with one another [19]. A clearer example of this can be seen in Figure 16, where aluminium was simulated on the left and wood on the right. There is a rapid temperature change when heat is transferred from aluminium to wood. The difference between wood simulated on the left and aluminium simulated on the right indicates that heat dissipates slowly from wood to aluminium, supporting the hypothesis that materials with a lower conductivity are poorer heat conductors.

3.4.2 Copper and styrofoam

The second test focused on copper and styrofoam. The thermal conductivity values used for copper were 413 W/m·K and 0.027 W/m·K for styrofoam, under the assumption that the ambient temperature is 300K [20,21]. For the second test involving these two materials, there were two simulations carried out with varying widths. In the first simulation, copper was simulated on the left and styrofoam on the right, separated on a left-right axis. Copper and styrofoam were simulated with widths of 0.8m and 0.2m, respectively. As part of this simulation, a fixed temperature of 500K is applied at the bottom boundary, convection at the top and left boundary, and insulation at the right boundary. Table 6 and Figure 17 illustrate the results of this simulation.

Table 6
Matlab numerical results of various grid sizes for Simulation 1

Grid Size	Iterations	Elapsed Time (s)
5×5	38	0.025315
9×9	112	0.028011
17×17	373	0.040246
33×33	1213	0.163970
65×65	3775	0.616496

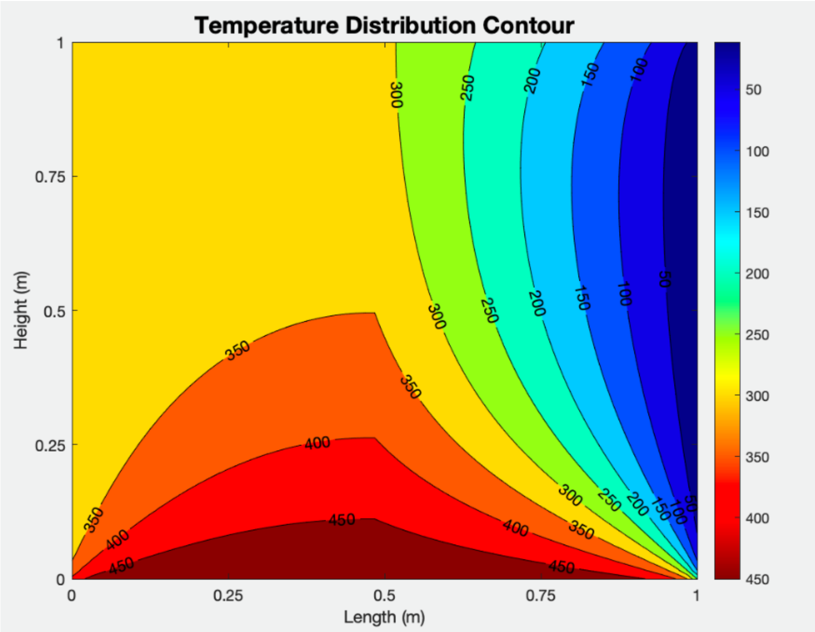

Fig. 17. 65×65 Grid for Copper (Left) and Styrofoam (Right) – Fixed Temperature (Bottom), Convection (Top and Left), Insulation (Right)

In the second simulation, styrofoam was simulated on the left and copper on the right, separated on a left-right axis. Styrofoam and copper were simulated with widths of 0.8m and 0.2m, respectively. In this simulation, the same boundary conditions are applied as in Simulation 1. Table 7 and Figure 18 illustrate the results of this simulation.

Table 7
Matlab numerical results of various grid sizes for Simulation 2

Grid Size	Iterations	Elapsed Time (s)
5×5	72	0.029423
9×9	160	0.028720
17×17	400	0.046871
33×33	1100	0.160142
65×65	3195	0.856505

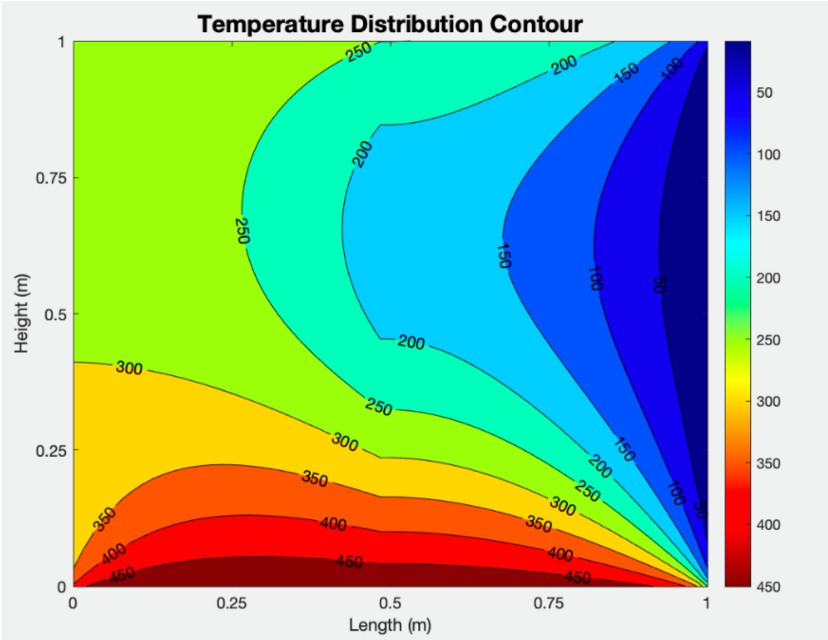

Fig. 18. 65×65 Grid for Styrofoam (Left) and Copper (Right) – Fixed Temperature (Bottom), Convection (Top and Left), Insulation (Right)

Based on the plots presented in Figure 17 and Figure 18, the temperature distribution did not appear to be similar. It is easy, however, to differentiate the boundary conditions at first glance. Clearly, convection occurs at the top and left boundaries, as indicated by the widely spaced contour lines, which indicate rapid changes in temperature. There is insulation at the right boundary, which has a fixed temperature of 300K to prevent heat transfer to the surrounding environment, as well as a fixed temperature at the bottom boundary that illustrates how 500K slowly dissipates heat across the boundaries. Styrofoam as an insulated conductor appears to show a gradual change in temperature by the closely spaced contour lines in Figure 17 when compared to the insulated boundary in Figure 18, with copper simulated on the right. Similarly, convection is more prominent in Figure 18, where there is a drastic temperature change at the left and top boundaries. It is also evident that when copper occupies a larger width of 0.8m in the first simulation, heat dissipates rapidly, as can be seen by the widely spaced contour lines extending from the bottom boundary. It appears that convection is causing heat dissipation to be slowed down in the same simulation, as the temperature is seen slowly decreasing towards the right insulated boundary.

3.4.3 Porcelain and granite

In previous sections, heat transfer has been examined across different combinations of materials. In order to gain a better understanding of how heat transfer occurs in our daily lives, practical applications have been used to study how realistic materials transfer heat. This section discusses underfloor heating in cold climates as an example. Essentially, underfloor heating entails warming up a home with flexible and strong tubing embedded in the floor. As an efficient and affordable means of providing thermal comfort, it is a superior alternative to radiators in many respects. When using underfloor heating, tile and stone are the most suitable flooring types. In order to study the heat transfer between porcelain and granite, which is commonly used in tiles and stones, simulations of these materials are being conducted.

For the simulation of porcelain and granite, the thermal conductivity values used for porcelain were 1.5 W/m·K and 3.1 W/m·K for granite, under the assumption that the ambient temperature is

300K [22,23]. As part of the current test involving these two materials, two simulations were performed. In the first simulation, porcelain was simulated on the left and granite on the right, separated on a left-right axis. Porcelain and granite were simulated with equal widths of 0.5m each. As part of this simulation, a fixed temperature of 320K is applied at the bottom boundary, convection at the top boundary, and insulation at the left and right boundary. Furthermore, the simulations were conducted with parameter values and boundary conditions that were as close to those in reality as possible. Generally, the floor heating system operates at a temperature of 45°C, which is approximately 320K [24]. Table 8 and Figure 19 illustrate the results of this simulation.

Table 8
Matlab numerical results of various grid sizes for Simulation 1

Grid Size	Iterations	Elapsed Time (s)
5×5	35	0.016309
9×9	98	0.022122
17×17	299	0.033567
33×33	954	0.123004
65×65	3074	0.434522

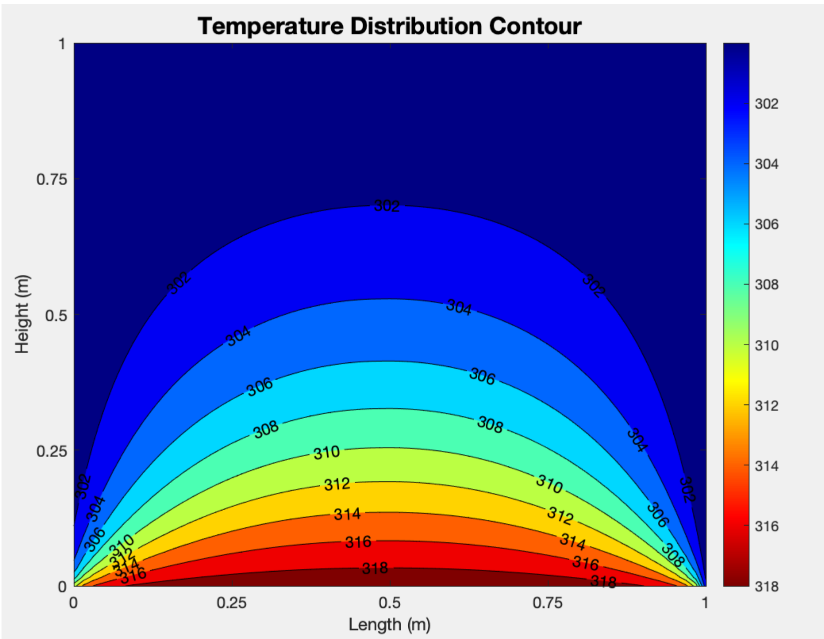

Fig. 19. 65×65 Grid for Porcelain (Left) and Granite (Right) – Fixed Temperature (Bottom), Convection (Top), Insulation (Left and Right)

In the second simulation, granite was simulated on the left and porcelain on the right, separated on a left-right axis. Granite and porcelain were simulated with equal widths of 0.5m each. In this simulation, the same boundary conditions are applied as in Simulation 1. Table 9 and Figure 20 illustrate the results of this simulation.

Table 9
Matlab numerical results of various grid sizes for Simulation 1

Grid Size	Iterations	Elapsed Time (s)
5×5	30	0.021680
9×9	900	0.020998
17×17	286	0.034629
33×33	932	0.105756
65×65	3039	0.380941

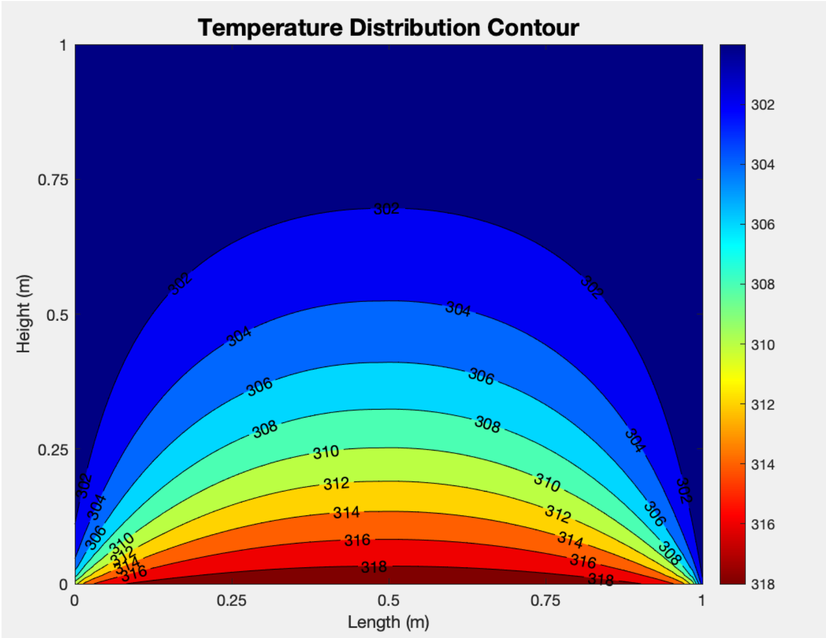

Fig. 20. 65×65 Grid for Granite (Left) and Porcelain (Right) – Fixed Temperature (Bottom), Convection (Top), Insulation (Left and Right)

It appears that there are no apparent differences between the two temperature distribution plots in Figure 19 and Figure 20, despite simulating the materials at different positions. As can be seen from the plots, heat is readily dissipated from the bottom boundary, where the temperature is fixed, to the top boundary, where it is insulated on the sides. The plots clearly depict how an underfloor heating system dissipates heat.

In general, heat flows from hotter to colder regions, and the ground temperature is usually lower than the temperature of the underground heating element. As a result of the convection that occurs at the top of the floor, the room can be heated evenly. Upon heating, the air near the floor rises, creating a natural convection current that circulates warm air throughout the space. In this manner, heat can be distributed evenly across the room, and hot spots and cold spots can be avoided. By insulating the sides of the underfloor heating system, the heat loss to the surrounding environment can be reduced. Thus, the underfloor heating system may be more efficient, and energy consumption may be reduced. Generally, underfloor heating systems are most effective when convection is applied to the top surface of the floor and insulation is applied to its sides, ensuring an even and efficient distribution of heat throughout the space.

4. Conclusions

The purpose of this study was to develop a computational code for solving two-dimensional steady-state heat conduction problems using Gauss-Seidel iterations. The developed code models heat transfer accurately across a variety of combinations of materials and allows users to input initial and boundary conditions to produce contour plots identical to those produced by ANSYS. By comparing analytical and numerical solutions obtained from Excel, MATLAB and ANSYS, the accuracy of numerical solutions for the Laplace equation was evaluated and verified. Results of this study have proven that solutions from Excel and MATLAB are just as accurate as costly programs like ANSYS. Furthermore, simulations with different grid sizes were conducted in order to determine the optimum grid size. Findings have suggested that once an ideal grid size has been determined, it is possible to save time by not having to run so many iterations. Analysis of the results has concluded

65×65 as the optimum grid size since it required a significantly shorter execution time for the code while achieving the same results as the analytical solution. To conclude, this study presents a method of calculating heat transfer by numerical means that is both accurate and efficient. As a result of the development of this code, it can be used in the analysis of two-dimensional heat transfer between different combinations of materials. On the basis of these findings, future studies can further optimize the code and increase the complexity of the simulations, such as simulating heat conduction through irregular shapes.

Acknowledgement
This research was not funded by any grant.

References

[1] Rafique, Muhammad Musaddique Ali. "Modeling and Simulation of Heat Transfer Phenomena." *Heat Transfer: Studies and Applications* 225 (2015). https://doi.org/10.5772/61029
[2] Li, Kun, Shiquan Shan, Qi Zhang, Xichuan Cai, and Zhijun Zhou. "A computational method to solve for the heat conduction temperature field based on data-driven approach." *Thermal Science* 26, no. 1 Part A (2022): 233-246. https://doi.org/10.2298/TSCI200822165L
[3] Matt, Carlos Frederico, and Manuel Ernani Cruz. "Heat conduction in two-phase composite materials with three-dimensional microstructures and interfacial thermal resistance." In *Heat Transfer in Multi-Phase Materials*, pp. 63-97. Berlin, Heidelberg: Springer Berlin Heidelberg, 2010. https://doi.org/10.1007/8611_2010_10
[4] Alfarawi, Suliman, Azeldin El-sawi, and Hossin Omar. "Exploring discontinuous meshing for CFD modelling of counter flow heat exchanger." *Journal of Advanced Research in Numerical Heat Transfer* 5, no. 1 (2021): 26-34.
[5] Karwa, Rajendra. *Heat and mass transfer*. Springer, 2020. https://doi.org/10.1007/978-981-15-3988-6
[6] Aly, Shahzada Pamir, Said Ahzi, Nicolas Barth, and Benjamin W. Figgis. "Two-dimensional finite difference-based model for coupled irradiation and heat transfer in photovoltaic modules." *Solar Energy Materials and Solar Cells* 180 (2018): 289-302. https://doi.org/10.1016/j.solmat.2017.06.055
[7] Muhammad, Nura Mu'az, Nor Azwadi Che Sidik, Aminuddin Saat, and Bala Abdullahi. "Effect of nanofluids on heat transfer and pressure drop characteristics of diverging-converging minichannel heat sink." *CFD Letters* 11, no. 4 (2019): 105-120.
[8] Böckh, Peter, and Thomas Wetzel. *Heat transfer: basics and practice*. Springer Science & Business Media, 2012.
[9] Hassan, Qais Hussein, Shaalan Ghanam Afluq, and Mohamed Abed Al Abas Siba. "Numerical investigation of heat transfer in car radiation system using improved coolant." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 83, no. 1 (2021): 61-69. https://doi.org/10.37934/arfmts.83.1.6169
[10] Hasan, Husam Abdulrasool, Zainab Alquziweeni, and Kamaruzzaman Sopian. "Heat transfer enhancement using nanofluids for cooling a Central Processing Unit (CPU) system." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 51, no. 2 (2018): 145-157.
[11] Tan, C., S. Zainal, C. J. Sian, and T. J. Siang. "ANSYS simulation for Ag/HEG hybrid nanofluid in turbulent circular pipe." *Journal of Advanced Research in Applied Mechanics* 23, no. 1 (2016): 20-35.
[12] AbdulWahid, Ammar F., Zaid S. Kareem, and Hyder H. Abd Balla. "Investigation of heat transfer through dimpled surfaces tube with nanofluids." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 67, no. 2 (2020): 116-126.
[13] Sidik, Nor Azwadi Che, Chow Hoong Kee, Siti Nurul Akmal Yusof, and Ahmad Tajuddin Mohamad. "Performance enhancement of cold thermal energy storage system using nanofluid phase change materials." *Journal of Advanced Research in Applied Mechanics* 62, no. 1 (2019): 16-32.
[14] Lee, Minhyung, Gwanyong Park, Changyoung Park, and Changmin Kim. "Improvement of grid independence test for computational fluid dynamics model of building based on grid resolution." *Advances in Civil Engineering* 2020 (2020): 1-11. https://doi.org/10.1155/2020/8827936
[15] Zulkurnai, Fatin Farhanah, Norhidayah Mat Taib, Wan Mohd Faizal Wan Mahmood, and Mohd Radzi Abu Mansor. "Combustion characteristics of diesel and ethanol fuel in reactivity controlled compression ignition engine." *Journal of Advanced Research in Numerical Heat Transfer* 2, no. 1 (2020): 1-13.
[16] Japar, Wan Mohd Arif Aziz, Nor Azwadi Che Sidik, Natrah Kamaruzaman, Yutaka Asako, and Nura Mu'az Muhammad. "Hydrothermal performance in the Hydrodynamic Entrance Region of Rectangular Microchannel Heat Sink." *Journal of Advanced Research in Numerical Heat Transfer* 1, no. 1 (2020): 22-31.

[17] Mahesh, M., P. Thangavel, M. S. Abdullah, B. Arulpriyan, P. Gokul, and D. Mahesh Kumar. "Performance improvement in thermal conductivity measuring apparatus using spiral heating coil." In *AIP Conference Proceedings*, vol. 2492, no. 1. AIP Publishing, 2023. https://doi.org/10.1063/5.0113149

[18] Mussa, Hadeel Mahmood, and Tawfeeq Wasmi M. Salih. "Thermal conductivity of wood-plastic composites as insulation panels: Theoretical and experimental analysis." *Építöanyag (Online)* 73, no. 2 (2021): 54-62. https://doi.org/10.14382/epitoanyag-jsbcm.2021.9

[19] Sidik, Nor Azwadi Che, Saidu Bello Abubakar, and Siti Nurul Akmal Yusof. "Measurement of Fluid Flow and Heat Transfer Performance in Rectangular Microchannel using Pure Water and Fe_3O_4-H_2O Nanofluid." *Journal of Advanced Research in Applied Mechanics* 68, no. 1 (2020): 9-21. https://doi.org/10.37934/aram.68.1.921

[20] Horton, W. Travis, and J. Clair Batty. "Use of high-thermal conductivity composites in cryogenic systems." In *Cryogenic Optical Systems and Instruments VII*, vol. 2814, pp. 217-227. SPIE, 1996. https://doi.org/10.1117/12.254145

[21] Srihanum, Adnan, Maznee TI Tuan Noor, Kosheela PP Devi, Seng Soi Hoong, Nurul H. Ain, Norhisham S. Mohd, Nik Siti Mariam Nek Mat Din, and Yeong Shoot Kian. "Low density rigid polyurethane foam incorporated with renewable polyol as sustainable thermal insulation material." *Journal of Cellular Plastics* 58, no. 3 (2022): 485-503. https://doi.org/10.1177/0021955X211062630

[22] Garcia, Eugenio, A. De Pablos, M. A. Bengoechea, L. Guaita, Maria Isabel Osendi, and P. Miranzo. "Thermal conductivity studies on ceramic floor tiles." *Ceramics International* 37, no. 1 (2011): 369-375. https://doi.org/10.1016/j.ceramint.2010.09.023

[23] Cho, W. J., S. Kwon, and J. W. Choi. "The thermal conductivity for granite with various water contents." *Engineering Geology* 107, no. 3-4 (2009): 167-171. https://doi.org/10.1016/j.enggeo.2009.05.012

[24] Vadiee, Amir, Ambrose Dodoo, and Elaheh Jalilzadehazhari. "Heat supply comparison in a single-family house with radiator and floor heating systems." *Buildings* 10, no. 1 (2019): 5. https://doi.org/10.3390/buildings10010005