Environment. Technology. Resources. Rezekne, Latvia
Proceedings of the 15th International Scientific and Practical Conference. Volume III, 79-83

Numerical Simulation and Analysis of Two-Dimensional Steady-State Heat Conduction in 2d Rectangular Domain

Lyuba Evtimova Gyurova
Naval Tactics Department
Nikola Vaptsarov Naval Academy
Varna, Bulgaria
l.gyurova@nvna.eu

***Abstract* — This work recommends the use of an algorithm to solve a number of engineering problems related to transient conduction during the various processes associated with heat transfer through a single or multilayer wall.**
A detailed study of the steady-state thermal conductivity in a two-dimensional rectangular domain is carried out. Using numerical methods specific to the solution of the heat conduction equations, a model has been used which gives accurate representations of the heat distribution in the given geometry.
The focus of this study is aimed at deepening the practical implications of the used numerical methodology. The results obtained from analysis highlight the potential for optimizing processes related to heat conduction in various engineering fields. The proposed numerical approach reveals opportunities for precise modelling and improvement of the thermal characteristics of various systems especially applicable in electronics, industrial systems and the implementation of modern thermal insulation materials.

Keywords* — *steady-state thermal conductivity, temperature, heat flux, CFD simulation

I. INTRODUCTION

In today's modern, constantly evolving world, the use of software products for numerical simulations leads to a better understanding and analysis of various physical processes. Heat conduction, as such a process, represents a micro-process of heat transfer in direct contact between the elements with different temperature that make up a given medium. This process is common in various engineering, technological, industrial, etc. areas and its optimization is essential. Examples of this are cooling, heating and air conditioning systems and their components such as heat exchangers and piping; tanks for various fluids and their accompanying pumps and throttling devices; thermal switch to control the thermal resistance; thermal insulation of walls and windows; and many others [1] for which a number of thermal problems can be solved.

The present work focuses on the research and performance of the numerical simulation and analysis of steady heat conduction within a two-dimensional rectangular domain, specifically using the ANSYS software [2]. By applying advanced computational methods, various problems related to heat conduction can be modeled more effectively.

The primary goal of this article is to ascertain the temperature distribution and heat flux within a 2D rectangular domain, while also investigating the impact of individual parameters on these thermal characteristics. In the given two-dimensional domain, the temperature changes in only two directions, x and y, and the steady-state thermal conductivity means that it does not change with time, $\frac{\partial}{\partial t} = 0$. This approach is suitable for understanding the basic laws of heat conduction in two-dimensional systems and subsequently developing more effective methods of management and control of processes related to heat transfer through different media [3], [4].

To solve the given physical problem, a mathematical model is used, in which Laplace's differential equation is solved for a two-dimensional flow in a rectangular region in the absence of heat generation [5], [6]. The mathematical model used is solved using the ASNYS solver, which uses the finite element method [7]. This involves drawing the geometry of the domain, setting the boundary conditions of the problem that define heat transfer between the object and its surroundings, generating a computational mesh with a specified set of elements to solve the heat transfer equations, and discretizing these differential equations to solve them by ANSYS solver, which will clarify the physics of the problem [8], [9]. After program solving for all the

Print ISSN 1691-5402
Online ISSN 2256-070X
https://doi.org/10.17770/etr2024vol3.8122

elements that make up the mesh, an analysis of the obtained results is made.

II. MATHERIALS AND METHODS

A. Problem specification

To fulfil the set objective of the study, a 2D rectangular area is drawn in ANSYS and a specific mathematical model is set. The mathematical model describing the heat exchange in a two-dimensional rectangular domain is introduced in ANSYS, where it is solved numerically. The model is a Laplace's equation for energy conservation (1).

$$k\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right)=0, \quad (1)$$

where: *k, W/m°C* – thermal conductivity coefficient;

T, °C – temperature of the domain.

The problem under consideration is a Boundary value problem related to 2D steady conduction, which means that predefined boundary conditions must be introduced. The domain has width $\Delta x=W$ and height $\Delta y=H$, where the height is twice the width, $H=2W$ [2].

When solving the given problem in ANSYS, it is necessary to make simplification by using dimensionless domain sizes and the mathematical model. In ANSYS, the dimensionless model (2) with boundary values is introduced, which corresponds to the dimensional one (1).

$$\left(\frac{\partial^2 \theta}{\partial x^2}+\frac{\partial^2 \theta}{\partial y^2}\right)=0. \quad (2)$$

Dimensioning is performed by defining the dimensionless coordinates $x^*\equiv\frac{x}{W}$; $y^*\equiv\frac{y}{W}$ и безразмерната температура $\theta\equiv\frac{T-T_\infty}{T_0-T_\infty}$ [2], където:

T_0, *°C* – constant temperature of the bottom face;

T_∞, *°C* – temperature of the fluid bathing the right face.

For the given domain, three boundary conditions are set for the four faces that must match the dimensionless model. The dimensionless and dimensional problems are given in the Fig. 1 and Fig. 2, respectively.

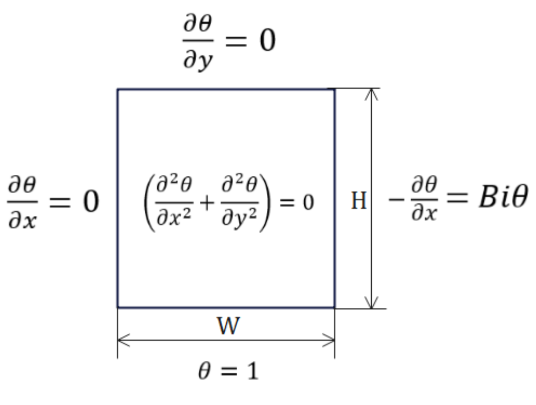

Fig. 1. Dimensionaless boundary value problem

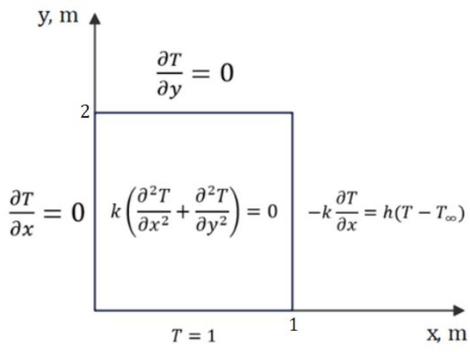

Fig. 2. Dimensional boundary value problem

- ✓ 1st boundary condition – along the lower isothermal surface the temperature must be kept constant. In order to fulfil this condition, a temperature $T=T_0$ is set in the dimensional equation on the lower side, where the dimensionless temperature $\theta=1$;
- ✓ 2nd boundary condition – no heat exchange with the environment must take place on the left and upper faces, the surfaces must be adiabatically insulated. This means that the heat flux must be zero in both directions, $\frac{\partial T}{\partial x}=0$, $\frac{\partial T}{\partial y}=0$ and *T* is assumed to be *θ*;
- ✓ 3rd boundary condition – presence of convection due to the fluid flowing around the wall on the right face of the domain, $-\frac{\partial\theta}{\partial x}=Bi\theta$. In the specific case, the Biot number is assumed to be $Bi\equiv\frac{hW}{k}=5$. In the mathematical model, the coefficients are set with values: $k=1$, $W=1$, $h=Bi$ and $T_\infty=0$. For the coefficient *k*, the condition of constant thermal conductivity in all directions is fulfilled. In the specific case, the importance falls on the convection coefficient *h* and its setting. It should be equal to *5*.

For more clarity on the use of the Biot number (Bi), some clarifications are provided. The Biot number is a critical dimensionless parameter used in the field of heat transfer, serving to correctly analyse the interaction between thermal conductivity inside a solid body and convection of its surface layer. In the sense of thermal resistance, the Biot number represents the ratio of the thermal conduction resistance inside the body to the convection resistance between the surface and the environment. Or put it in another way, the value of the Biot number is a measure of the temperature drop in the solid relative to the temperature difference between the surface and the liquid and is defined with a limit value of *1*. At $Bi>1$, the temperature change inside the body takes place much more slowly than that between its surface layer and the fluid. Conversely, for $Bi<1$ the body has a smaller temperature gradient compared to that between the surface and the liquid, and heat will flow much faster inside the material than away from its surface. This would allow an assumption to be made of a uniform temperature distribution in the material [3], [10].

B. Mesh generation

In any problem involving heat exchange, it is important to know the distribution of heat throughout the considered domain and its direction depending on the set boundary conditions. The heat flux in any direction is proportional to the temperature gradient in that direction. This means, in the present case for example, that if the flow reaches the left adiabatic face where there is no heat exchange, there will also be no temperature gradient in the x direction out of the domain, $q_x=0$.

Discretization of the rectangular domain is done by setting a geometry generation mesh containing 32 cells and 121 nodes, Fig. 3.

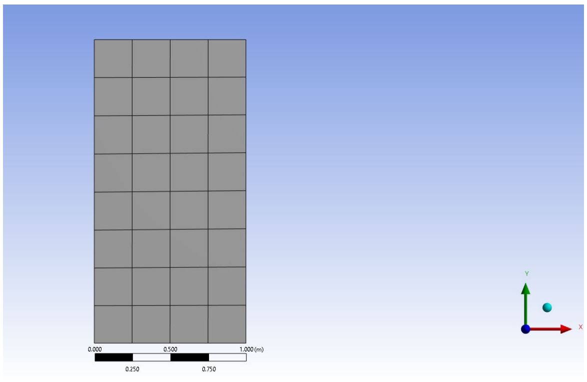

Fig. 3. View of mesh geometry

All elements have four nodes each, and each internal node is common to the four elements surrounding it. In order to determine the temperature of the elements, the ANSYS solver uses a bilinear polynomial interpolation of the four nodal temperatures of a cell, thereby obtaining a weighted average value of these four values. For all nodal temperatures, the ANSYS solver constructs a system of algebraic equations, and each algebraic equation relates a nodal temperature to the elements around the corresponding node. Thus, the value of this node is related to all other temperature values in the surrounding elements. In this way, the temperature everywhere in the domain can be determined, by interpolation from nodal temperatures, and hence the heat flux obtained.

To reduce the calculation error, it is necessary to increase the number of elements, which increases the accuracy of the mesh. In this way, the number of equations describing the nodal temperatures will also increase. Another way to achieve error reduction is by increasing the order of the polynomial in each element. This can be done when increasing the nodes for each element, i.e. in addition to the four end nodes, there should be one more between every two, and the ANSYS solver will again generate algebraic equations and calculate the temperature in all of them. Then the interpolation will be of second order in both directions, which will lead to a better and more accurate solution to the problem.

III. Results and Discussion

A. Change in temperature in the researched domain

The temperature can be traced at any point in the domain, as well as along a straight line in both directions, and an assessment can be made as to whether the set boundary conditions are met.

Fig. 4 shows the temperature in the entire domain within limits $T=(1\div0.045)$ ºC. On the lower surface, the temperature is unchanged, which shows that the boundary condition for isothermality on this face is met.

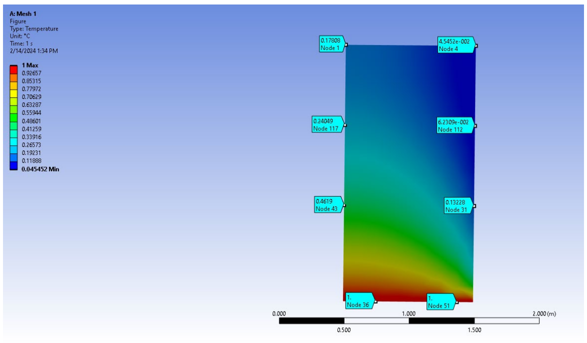

Fig. 4. Change in temperature in the domain

Tracing the temperature at 51 points along a straight horizontal line, Fig. 5, in any other section of the area, it becomes clear that it decreases from right to left, which is due to the fact that there is no heat exchange with the environment through the left face, and such takes place only through the right face between the solid body and the fluid flowing around it. This can also be seen from the graphical representation of the curve showing the variation of temperature along length, $\frac{dt}{dx}$. Towards the adiabatic boundary condition, the slope of the curve is zero and the temperature is almost constant and highest. Further approaching the right face, the slope becomes greater, the temperature decreases, which is due to the convection between the fluid and the wall.

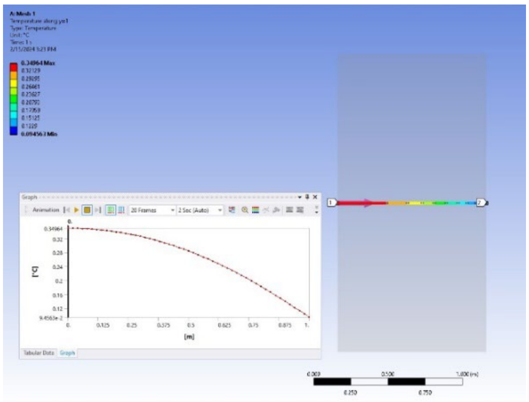

Fig. 5. Temperature variation along a line

Table 1 gives the value of the temperature variation.

Table 1 Temperature values along a line

Point	W, [m]	T, [ºC]	Point	W, [m]	T, [ºC]
1	0	0.34964	26	0.5	0.28412
2	0.02	0.34954	27	0.52	0.2787
3	0.04	0.34923	28	0.54	0.27309
4	0.06	0.34871	29	0.56	0.2673
5	0.08	0.34798	30	0.58	0.26131
6	0.1	0.34704	31	0.6	0.25514
7	0.12	0.34589	32	0.62	0.24877
8	0.14	0.34453	33	0.64	0.24222
9	0.16	0.34296	34	0.66	0.23548

10	0.18	0.34118	35	0.68	0.22855
11	0.2	0.33919	36	0.7	0.22143
12	0.22	0.33699	37	0.72	0.21412
13	0.24	0.33459	38	0.74	0.20662
14	0.26	0.33197	39	0.76	0.19884
15	0.28	0.32913	40	0.78	0.19083
16	0.3	0.32609	41	0.8	0.1827
17	0.32	0.32284	42	0.82	0.17444
18	0.34	0.31938	43	0.84	0.16606
19	0.36	0.3157	44	0.86	0.15756
20	0.38	0.31182	45	0.88	0.14893
21	0.4	0.30773	46	0.9	0.14018
22	0.42	0.30343	47	0.92	0.1313
23	0.44	0.29891	48	0.94	0.1223
24	0.46	0.29419	49	0.96	0.11318
25	0.48	0.28926	50	0.98	0.10393
			51	1	0.0946

These findings clearly indicate that the set boundary conditions are accurately defined and the simulation results confirm them.

B. Distribution of heat flux in the domain

Fig. 6 shows the heat flux distribution from each node in the form of vectors passing through the domain. The distribution is two-dimensional and the two components q_x and q_y are related to the temperature gradients for the two directions respectively.

Analysis of the lower isothermal face shows that there is no heat flow there and the heat vector is perpendicular only in the *y* direction. The most heat losses are observed in the lower right, and for this reason a large amount of heat must be supplied to compensate them and thus keep the temperature constant.

On the left and upper faces, the heat vectors are parallel in the respective directions, which confirms the absence of heat exchange across these boundaries. However, the heat flux is not strictly zero, it has a very small value, $\vec{q} = 0.00052415\ W/m^2$ and the variation is only in one direction. Of interest is the upper left corner where the heat vector is not perpendicular to either the *x* or *y* direction. This is due to the fact that there is no heat flow in either direction in this part of the area and therefore the heat vectors change direction.

On the right side, heat exchange takes place between the wall and the fluid and the heat flows to the fluid. At a height, heat exchange decreases due to the smaller temperature difference.

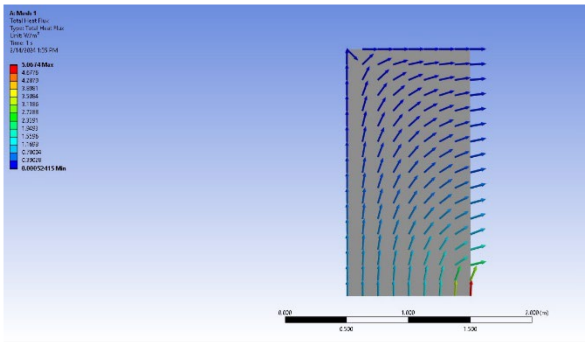

Fig. 6. Heat flux distribution in the domain

C. Checking the results obtained

Since the inputted mathematical model is subject to the principle of energy conservation, it is necessary to check whether it is respected. The check is made for the heat flux passing through the boundaries per unit wall thickness.

Fig. 7 shows the heat calculated by the ANSYS solver along the lower isothermal surface, $q = 1.6701\ W$. It is positive because its direction is toward the domain, Fig. 6.

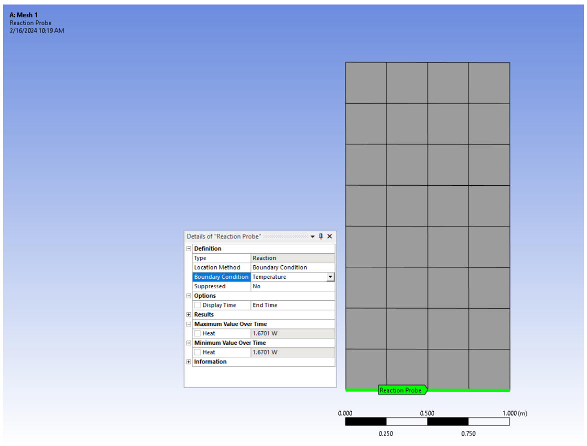

Fig. 7. Heat transfer along the isothermal surface

It can be seen from Fig. 8 that due to the presence of heat transfer through the right boundary where convection takes place, heat is transferred to the flowing fluid and it is negative with a value of $q = -1.6701\ W$.

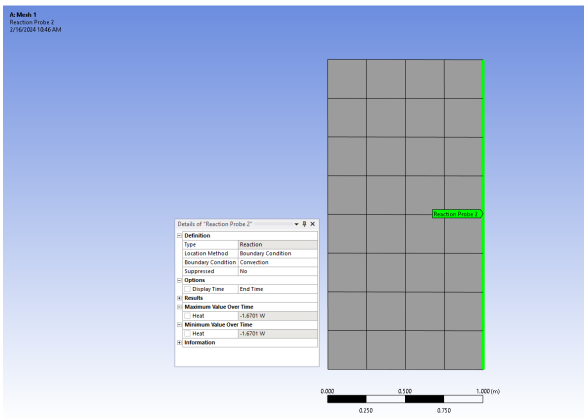

Fig. 8. Heat transfer on the convective surface

In the specific case, in the absence of heat exchange through the other two boundaries and due to the condition that there is no heat generation, the heat balance depends only on the heat exchange through the lower and right surface of the region, and from the obtained values it can be seen that it is achieved.

IV. CONCLUSION

Numerical simulation and analysis of two-dimensional steady-state heat conduction in a rectangular domain using ANSYS proves to be extremely informative in studying thermal behaviour and optimizing design for efficient heat transfer.

In the present study, only the theoretical justification of the given problem is considered, without the presence of experimental data. As a future project, it is necessary to create a specialized experimental setup to provide an opportunity to compare the experimental data with the theoretically obtained ones. Moreover, when real problems arise where heat exchange is present, this approach turns out to be good enough and can be applied, as well as serve to solve similar problems in three-dimensional domains or include more complex boundary conditions for more further increasing the accuracy of the simulations.

Understanding two-dimensional heat conduction in rectangular domains is crucial for:

- Optimizing cooling systems in electronics;
- Improving insulation efficiency in buildings;
- Enhancing automotive cooling systems;
- Optimizing industrial processes for energy efficiency;
- Designing thermal management solutions in aerospace.

These applications demonstrate the importance of heat conduction analysis in various engineering fields for optimizing designs and improving performance.

REFERENCES

[1] Two-Dimensional, Steady-State Conduction https://www.cambridge.org/tn/files/6213/6698/3410/2.pdf

[2] Cornell University. (Dec 19. 2018). 2D Steady Conduction – Mesh. Simulation Wiki. https://confluence.cornell.edu/display/SIMULATION/2D+Steady+Conduction+-+Mesh

[3] T. L. Bergman, A. S. Lavine, F. P. Incropera and D. P. DeWitt, Introduction to Heat Transfer, Ch. 3 One-Dimensional Steady-State Conduction and Ch. 4 Two-Dimensional Steady-State Conduction, Copyright 2011, 2007, 2002 by John Wiley & Sons, Inc., 6th Edition, 2011, Pages 111-376, ISBN 13 978-0470-50196-2.

[4] T. L. Bergman, A. S. Lavine, F. P. Incropera and D. P. DeWitt, Fundamentals of Heat and Mass Transfer, Copyright 2011, 2007, 2002 by John Wiley & Sons, Inc., 7th Edition, 2011, ISBN 13 978-0470-50197-9

[5] W. H. David and Özisik M. N, Heat Conduction, John Wiley & Sons, Inc., Hoboken, New Jersey, 3rd Edition, 2012, ISBN 9780470902936.

[6] A. Bejan and A. D. Kraus, Heat Transfer Handbook, John Wiley & Sons, Inc., Hoboken, New Jersey, 3rd Edition, 2003, ISBN 978-0471321719.

[7] J. N. Reddy and D. K. Gartling, The Finite Element Method in Heat Transfer and Fluid Dynamics, CRC Press, Taylor & Francis Group, 3rd Edition, 2010, ISBN 9781420085983.

[8] S. Patankar, Numerical Heat Transfer and Fluid Flow, CRC Press Taylor & Francis Group, 1980, DOI https://doi.org/10.1201/9781482234213, eBook ISBN 9781315275130.

[9] Y. Jaluria, K. Torrance, Computational Heat Transfer, Published December 19 by CRC Press, 2nd Edition, 2002, ISBN 9781560324775.

[10] T. W. Davies, "BIOT NUMBER", THERMOPEDIA™ A-TO-Z GUIDE TO THERMODYNAMICS, HEAT & MASS TRANSFER, AND FLUIDS ENGINEERING, February 2011, DOI: 10.1615/AtoZ.b.biot_number.