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

II. MATERIALS AND METHODS

A. Problem specification

To fulfill 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).

\[ k \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right) = 0, \]  

(1)

where: \( k \), W/m°C – thermal conductivity coefficient; \( T \), °С – 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, \( H = 2W \).

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).

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

(2)

Dimensioning is performed by defining the dimensionless coordinates \( x' \equiv \frac{x}{W} \) and \( y' \equiv \frac{y}{W} \) and the dimensionless temperature \( \theta \equiv \frac{T-T_\infty}{T_0-T_\infty} \), where:

\( T_\infty \), °С – constant temperature of the bottom face;
\( T_\infty \), °С – 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. Dimensionless boundary value problem](image1)

![Fig. 2. Dimensional boundary value problem](image2)

A. 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](image)

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
odal temperature to the elements around the
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 \( T = (1 + 0.045) \) °C. 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](image)

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, \( \Delta T/\Delta x \).

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
convexion between the fluid and the wall.

![Fig. 5. Temperature variation along a line](image)

Table 1 gives the value of the temperature variation.

<table>
<thead>
<tr>
<th>Point</th>
<th>( W, [m] )</th>
<th>( T, [{\degree C}] )</th>
<th>Point</th>
<th>( W, [m] )</th>
<th>( T, [{\degree C}] )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>0.34064</td>
<td>6</td>
<td>0.08</td>
<td>0.34798</td>
</tr>
<tr>
<td>2</td>
<td>0.02</td>
<td>0.34054</td>
<td>7</td>
<td>0.12</td>
<td>0.34589</td>
</tr>
<tr>
<td>3</td>
<td>0.04</td>
<td>0.34023</td>
<td>8</td>
<td>0.14</td>
<td>0.34453</td>
</tr>
<tr>
<td>4</td>
<td>0.06</td>
<td>0.34871</td>
<td>9</td>
<td>0.16</td>
<td>0.34296</td>
</tr>
<tr>
<td>5</td>
<td>0.08</td>
<td>0.34758</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>0.1</td>
<td>0.34704</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>0.12</td>
<td>0.34589</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>0.14</td>
<td>0.34453</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>0.16</td>
<td>0.34296</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
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, $\bar{q} = 0.00052415 \text{ W/m}^2$ 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.

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, $q = 1.6701 \text{ W}$. It is positive because its direction is toward the domain, Fig. 6.

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 $q = -1.6701 \text{ W}$.

---

| 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  |

---

Fig. 6. Heat flux distribution in the domain

Fig. 7. Heat transfer along the isothermal surface

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