NUMERICAL STUDY OF NATURAL CONVECTION IN AN ENCLOSURE WITH DISCRETE HEAT SOURCES ON ONE OF ITS VERTICAL WALLS

by

Mehmet Turgay PAMUK*

Bahcesehir University, Besiktas, Istanbul, Turkey

Original scientific paper
https://doi.org/10.2298/TSCI190816448P

In this study, natural convection in a fluid-filled rectangular enclosure is analyzed using COMSOL© commercial software. The fluid in which natural convection takes place is a dielectric liquid called FC-75. Attached to one of the vertical walls of the enclosure is an array of rectangular protrusions, each representing computer chips mounted on a printed circuit board. The nominal power consumed by each chip is assumed to be 0.35, 1.07, 1.65, and 2.35 W. This corresponds exactly to the values used in the experiments, which were performed once by the author of this study. The results of the experiment and the numerical study are shown as Nusselt numbers vs. Rayleigh numbers, both being the most important dimensionless parameters of natural convection. A comparison of the results has shown that COMSOL© can achieve reliable results in similar problems, eliminating the need to build expensive experimental set-ups and spending time conducting experiments. The simulation results are aimed to be used in similar designs of electronic circuits in confined spaces.

Key words: *heat transfer, natural convection, enclosure, FC-75*

Introduction

Natural convection in enclosures has been extensively studied over the last 30 years, as it has many uses, *e. g.*, electronic circuits cooling, household refrigerator design, and electrical cabinet design. However, these studies may have significant deviations from one another due to experimental uncertainties in experiments or the application of the numerical method. On the other hand, analytic solutions have their weaknesses due to the assumptions made in the 2-D domain, since momentum equations coupled with energy equations are too difficult to solve in the 3-D domain. Faghri *et al.* [1] explained in their book various aspects of natural convection in enclosures. Enclosures are finite spaces bounded by walls and filled with fluids. Natural convection in enclosures, also called internal convection, takes place in rooms and buildings, ovens, cooling towers and electronic cooling systems. Internal convection, where a heated or cooled wall is in contact with the quiescent fluid and the boundary-layer can be fully developed. The internal convection cannot normally be treated with a simple boundary-layer theory, since all the fluid in the enclosure is involved in the convection. Incropera [2] laid the foundations for maintaining the best possible thermal environment in electronic enclosures and provided a comprehensive overview of convection cooling options. According to the author, the engineer

* Author's e-mail: turgaypamuk@hotmail.com

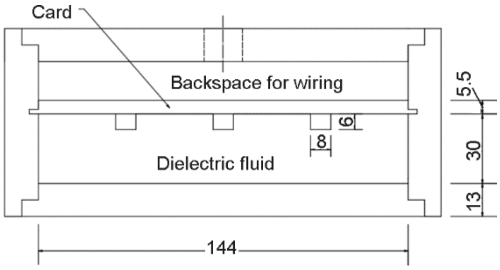

Figure 1. Top view of enclosure used in experiments [3]

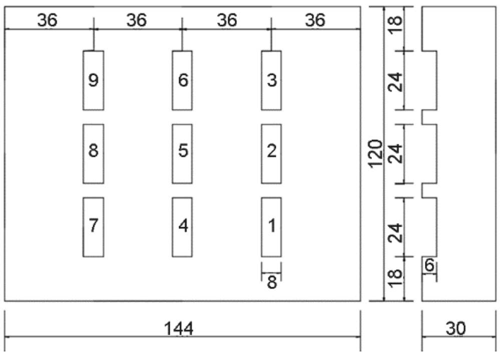

Figure 2. Front and side views of enclosure used in experiments [3]

must set the most efficient way for heat transfer from the electronic devices to an external coolant. Depending on the size and speed of the electronic circuits, as well as constraints due to non-thermal considerations, the coolant may be a gas or a liquid and the heat transfer may be by natural, forced or mixed convection or in the case a liquid, by pool or forced convection cooking. Pamuk [3] conducted a series of experiments with a wide range of power ratings in his M. Sc. thesis. The experimental set-up used in his dissertation is used as the basis for this study and is shown in figs. 1 and 2.

The enclosure has internal dimensions of 144 mm, 30 mm, and 120 mm in width, depth, and height. The vertical walls of the enclosure used in the construction are made of Plexiglass©, a transparent and low thermally conductive solid. The outer walls have a thickness of 13 mm and the wall on which a 3×3 array of protrusions (simulated chips) is mounted is 5.5 mm thick. The protrusions are made of aluminum with dimensions of 8 mm, 6 mm, and 24 mm as width, depth, and height, respectively. Foil type electrical heating elements are mounted behind these blocks using a high thermal conductivity bonding material. The power delivered by these aluminum blocks into the fluid is used as the net power, calculated from the electrical input (*IV*). The heat transfer medium used in the experiments is a chemical called FC-75, the properties of which are calculated using the polynomials provided in the thesis study. The top and bottom of the enclosure are aluminum heat exchangers with cooling water of 20 °C supplied by a constant temperature circulator to absorb the heat dissipated in FC-75. The raw temperature data was collected throughout the experiments by a data acquisition system along with the heat fluxes and the temperature dependent fluid properties calculated accordingly. The collected data was reduced and presented as Nusselt numbers *vs.* Rayleigh numbers, with a large number of calculations made using a computer program. The uncertainty analysis of the experiments was done for Nusselt and Rayleigh numbers and they were found to be 3.6% and 2.9%, respectively.

In his M. Sc. thesis, Aytar [4] examined various aspects of the same enclosure problem, such as variations in enclosure width and Prandtl number. He came up with Nusselt number correlations as functions of the flux-based Rayleigh numbers for each row of chips. An averaged Nusselt general correlation as a function of the Rayleigh-based flux yielded results within 10% of the experimental results. Gunes [5] numerically investigated 3-D buoyancy-driven convective flows and heat transfer for a cubic enclosure and vertical channels with spatially periodic, flush mounted, and protruding heat sources in his M. Sc. thesis. He found that protruding heat sources are better suited for electronic cooling than flush-mounted sources. He also concluded that 2-D and 3-D calculations for the vertical channel with protruding strip heat sources basically have the same convection cooling rate. Hermes *et al.* [6] implemented the stream function vorticity formulation in a CFD code for the analysis of air flow patterns and temperature distributions in closed cavities, particularly refrigerators and freezers. Their model is based on the principles of mass, momentum, and energy conservation. The flow was treated as incompressible and the buoyancy conditions were modeled using the Boussinesq approximation. They used a finite-volume method and an iterative method to discretize and solve the set of PDE. They reported comparisons with experimental data. Basak *et al.* [7] conducted a numerical study to investigate the steady laminar natural convection flow in a square cavity with uniformly and unevenly heated bottom wall and an adiabatic ceiling wall that maintains a constant temperature of the cold vertical walls. Their numerical method used in the present study provided satisfactory results for a range of Rayleigh numbers between 1000 and 100,000 and Prandtl numbers between 0.7 and 10. Turan *et al.* [8] carried out 2-D stationary simulations of laminar natural convection in square enclosures with differently heated sidewalls exposed to constant wall temperatures. In their study, it was assumed that the enclosures were completely filled with non-Newtonian fluids. They proposed new correlations for the average Nusselt number for both Newtonian and non-Newtonian power law fluids. Bazylak *et al.* [9] presented a computational analysis of heat transfer through an array of distributed heat sources on the bottom wall of a horizontal enclosure in their article. They modeled the heat sources as flush-mounted sources with given heat flux boundary conditions. They found that the source spacing being equal to the source length ensures effective convective heat transfer, and further increasing the source pitch does not lead to significant improvements. Varol *et al.* [10] numerically analyzed the natural convection heat transfer from a protruding heater in a triangular enclosure. The heater with a constant temperature boundary condition has a higher temperature than the inclined boundary of the triangle and the remaining walls are insulated. They formulated the study with regard to the vorticity stream function method and the numerical solution was performed with the finite difference method. Their working medium was air with Pr = 0.71. The results, presented in streamlines, isotherms, velocity profiles, and local and average Nusselt numbers, show that all parameters related to the geometric dimensions of the heater are effective for temperature distribution, flow field, and heat transfer. Their results also show that the Nusselt number with increasing Rayleigh number, size and position of the heater and the aspect ratio of the enclosure are important parameters for enhanced heat transfer. Doghmi *et al.* [11] performed a computational study to investigate the mixed convection in a 3-D ventilated cavity discretely heated from the side using the finite volume model for different governing parameters (Reynolds number, Richardson number, and heating section dimensions). They found that the flow structures and temperature distribution are considerably affected by the interaction between the inertia and the buoyancy forces. Purusothaman *et al.* [12] numerically investigated the buoyancy induced natural convection in a 3-D enclosure filled with composition of two different base fluids with dispersed Cu-nanoparticles. On one of the vertical walls of the enclosure is an array of 3×3 array of non-protruding (embedded) discrete heat sources. They found that by increasing solid volume fraction of the nanoparticles and fixing the aspect ratio at 2.0, the maximum heat transfer rate can be obtained.

Although some of the cited literature present the results of their numerical work, no validation using experiments is mentioned. On the other hand, this numerical study presented herein simulates the experiment under the best realistic conditions possible. Possible sources of the deviations between the experimental and numerical results are explain in the conclusion section.

Numerical model and theory

COMSOL© treats steady-state CFD problems in three distinct steps.

Meshing. The physics-controlled mesh is utilized in the CFD module for fluid-flow problems in order to compute accurate solutions. A boundary-layer mesh is generated in order to resolve the gradients.

Discretization. The fluid-flow physics interfaces use a Galerkin least-squares method to discretize the flow equations and generate the numerical model in space (2-D, 2-D axi-symmetry, and 3-D). The test function is designed to stabilize the hyperbolic terms and the pressure term in the transport equations. Shock-capturing techniques further reduce spurious oscillations. Additionally, discontinuous Galerkin formulations are used to conserve momentum, mass, and energy over internal and external boundaries.

Solvers. The flow equations are usually highly non-linear. In order to solve the numerical model equations, the automatic solver settings select a suitable damped Newton method. The methods are accelerated by state-of-the-art algebraic multigrid or geometric multigrid methods specifically designed for transport problems

Using the same geometry, fluid, array layout and net power ratings as those mentioned in the introduction, made it possible to compare the results of experiments with those of numerical study. Heat loss to the environment is small due to the negligible heat conduction through the outer vertical walls with small thermal conductivity, also due to the bulk fluid temperature inside the enclosure being very close to that of environment, allowed the vertical walls to be modelled with insulated boundary conditions, except the back of the heaters where a heat loss was taken into consideration in order to calculate the net heat input into the enclosure. This assumption simplifies the numerical method and reduces the computation time substantially. The calculation domain is meshed with 8732 triangular elements, 900 edge elements, and 80 vertex elements. The geometry of the model and the mesh is shown in fig. 3. It should be noted that there is only one domain, which is the fluid itself. Because all vertical surfaces are insulated walls and the lower and upper (horizontal) surfaces are isothermal walls, the model is therefore defined as a single fluid domain. Heat transfer, coupled with fluid-flow, is used with temperature dependent fluid properties as described above. Four sets of results are obtained for four different power settings. The temperature data along with the data of heat flow and fluid properties are used to calculate Nusselt numbers and Rayleigh numbers in the same way in the experiments.

When an incompressible steady-state flow is assumed, the heat transfer equations to be solved in the fluid region are:

$$\rho C_p \ddot{\mathrm{u}} \nabla T + \nabla q = 0 \tag{1}$$

$$q = -k \nabla T \tag{2}$$

Fluid-flow equations to be solved in the fluid domain are:

$$\rho(\ddot{\mathrm{u}}\nabla)\ddot{\mathrm{u}} = \nabla\{-pI + \mu[\nabla\ddot{\mathrm{u}} + (\nabla\ddot{\mathrm{u}})^T]\} + \vec{\mathrm{F}} \tag{3}$$

$$\rho\nabla(\ddot{\mathrm{u}}) = 0 \tag{4}$$

Previous equations are convective heat transfer, eq. (1), heat transfer at the solid-fluid interface, eq. (2), momentum, eq. (3), and continuity, eq. (4), with boundary conditions:

– $q = Q/A$ at the chip surfaces (surface heat flux boundary condition),

– $q = 0$ at vertical walls (insulated surface boundary condition),
– $T = T_c$ at bottom and top (horizontal) surfaces (isothermal surface boundary condition), and
– $\vec{u} = 0$ at all surfaces (no slip-wall boundary condition).

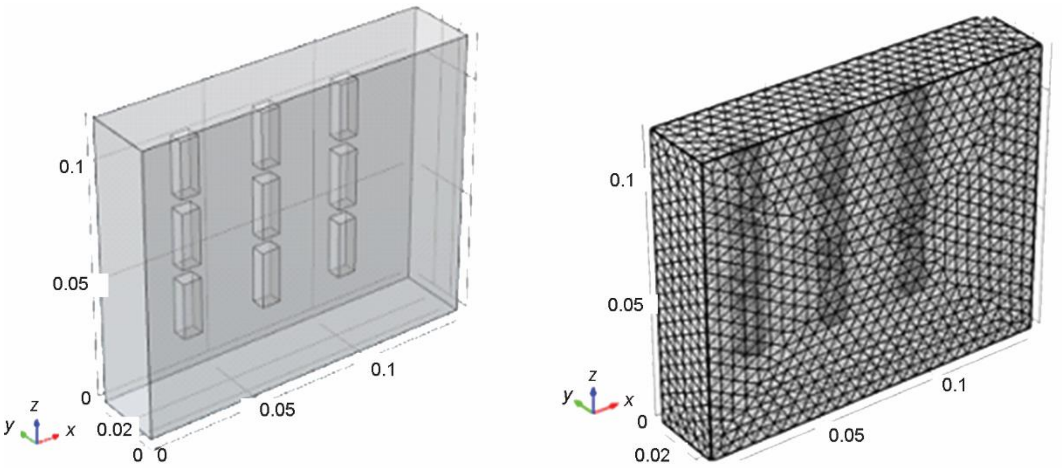

Figure 3. Geometry and the meshing created in COMSOL©

Numerical solutions

The total computation time on an Intel (R) Core (TM) i3-2350M processor at 2.30 GHz, a 2-core computer running Windows 8, is 1 hour and 36 minutes. For a faster convergence, the initial values of the temperature within the domain should be given with caution, otherwise convergence may not be achieved. It is recommended to start with coarser meshes to get an idea of what the solution might look like, then finer mesh sizes can be used to improve the solution. With further fine grids, no further improvement is achieved, because of the run time for the convergence becoming too much as well as too many iterations.

The surface temperature plots are shown in fig. 4. The area averaging module in COMSOL© is also used to calculate the area weighted average surface temperatures used in heat transfer calculations. Figures 5-7 show the temperature distributions away from the surface of the chips. It should be noted that the distance from the chip where the temperature levels are leveled off is the thickness of the thermal boundary-layer. It is also worth noting that the temperature gradient on the wall becomes zero when the wall is insulated, so that there is no expected outward heat flow. The pressure contours are shown in fig. 8, which are substantially the same as the hydrostatic pressure defined as $P = \rho g(H - y)$. Figure 11 shows velocity distributions in vertical planes passing through the centers of the chips for each column for the power consumption of 0.35 W. As can be seen from the plots, the velocity distribution is the same for each column. Therefore, only the velocity distributions of the middle column are displayed for other power settings, fig. 9. Similarly, velocity magnitudes away from the chips are shown in figs. 10-12 to obtain a complete picture of the physical phenomena. As with thermal boundary-layers, the distances from the chip surfaces where the velocities become zero represent the thickness of the momentum boundary-layer. It should be noted that these are velocity magnitudes and therefore the flow near the insulated wall actually is downward.

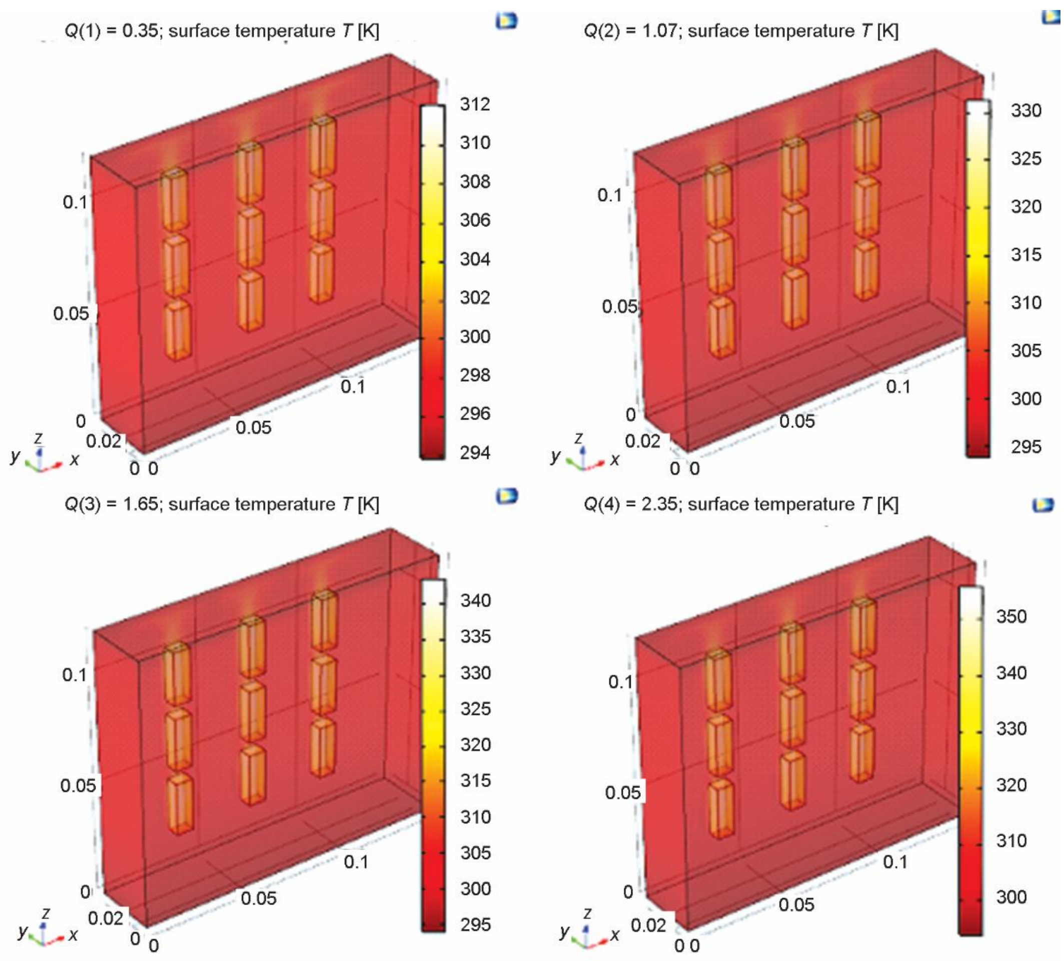

Figure 4. Temperature distribution for different power settings
(*for color image see journal web site*)

Heat transferred from each chip can be calculated using Newton's Law of Cooling:

$$q = h(T_s - T_c) \tag{5}$$

wherefrom:

– Nusselt number

$$\mathrm{Nu} = \frac{hL}{k} \tag{6}$$

– Grashof number

$$\mathrm{Gr} = \frac{g\beta L^3 \Delta T}{\nu^2} \tag{7}$$

– Rayleigh number

$$\mathrm{Ra} = \mathrm{Gr}\,\mathrm{Pr} \tag{8}$$

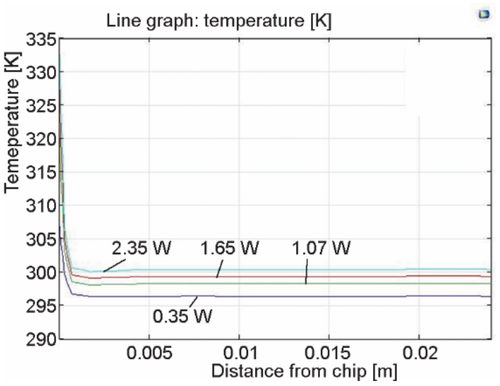

Figure 5. Temperature distributions away from the surface of bottom chip

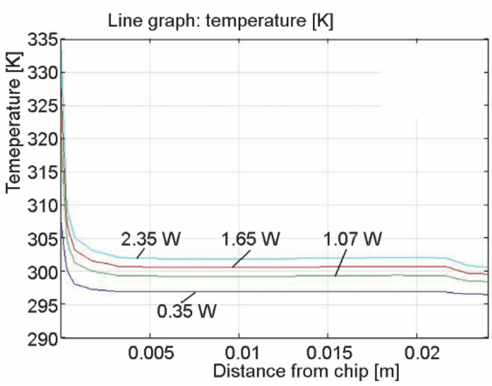

Figure 6. Temperature distributions away from the surface of middle chip

Using eqs. (6)-(8), Nusselt numbers and Rayleigh mumbers are calculated for bottom, middle and top rows separately. The fluid properties are calculated at film temperatures as $T_f = (T_s + T_c)/2$ for each chip. Table 1 shows the Rayleigh number dependence of the Nusselt number for both experiments and numerical calculations. As can be seen from the table, the results of the experimental and numerical study are substantially close to each other, taking into account the uncertainties of the experiments and assumptions in the numerical study. The difference is believed to have taken place due to the The Boussinesq approximation assumed by COMSOL© solver.

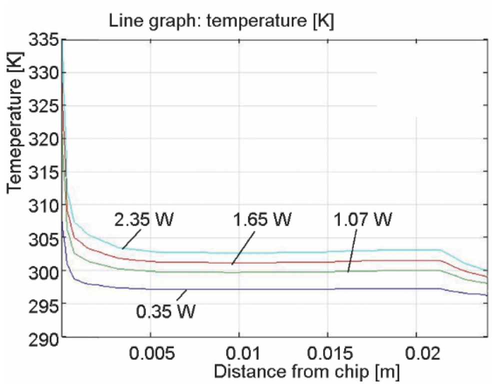

Figure 7. Temperature distributions away from the surface of top chip

A sample calculation is:

– Chip #5 (middle row): $Q = 1.07$ W, $T_s = 48.5\,°C$, $T_c = 20.7\,°C$, then $T_f = (T_s+T_c)/2 = 34.58\,°C$.
– Properties: $k = 0.0623$ W/(mK), $\rho = 1739.9$ kg/m^3, $\beta = 0.0013$ K^{-1}, $\nu = 7.36 \cdot 10^{-7}$ m^2/s, Pr = 21.82.
– Calculations: $h = 60.21$ W/(m^{2}K), Nu = 23.21, Gr = $9.83 \cdot 10^6$, and Ra = $2.15 \cdot 10^8$.

Table 1. Comparison of Nusselt numbers

	Experiment			COMSOL			Difference [%]		
Ra	Bottom	Middle	Top	Bottom	Middle	Top	Bottom	Middle	Top
$5.0 \cdot 10^7$	25.3	18.7	16.4	18.0	12.2	11.1	–29%	–35%	–32%
$1.0 \cdot 10^8$	32.3	22.0	19.1	26.5	16.4	15.4	–18%	–25%	–20%
$2.5 \cdot 10^8$	41.3	28.7	24.4	46.0	26.2	25.1	11%	–9%	3%
$3.0 \cdot 10^8$	40.3	30.0	25.1	50.5	28.4	27.4	25%	–5%	9%
$3.3 \cdot 10^8$	38.7	30.5	25.3	52.7	29.5	28.5	36%	–3%	12%

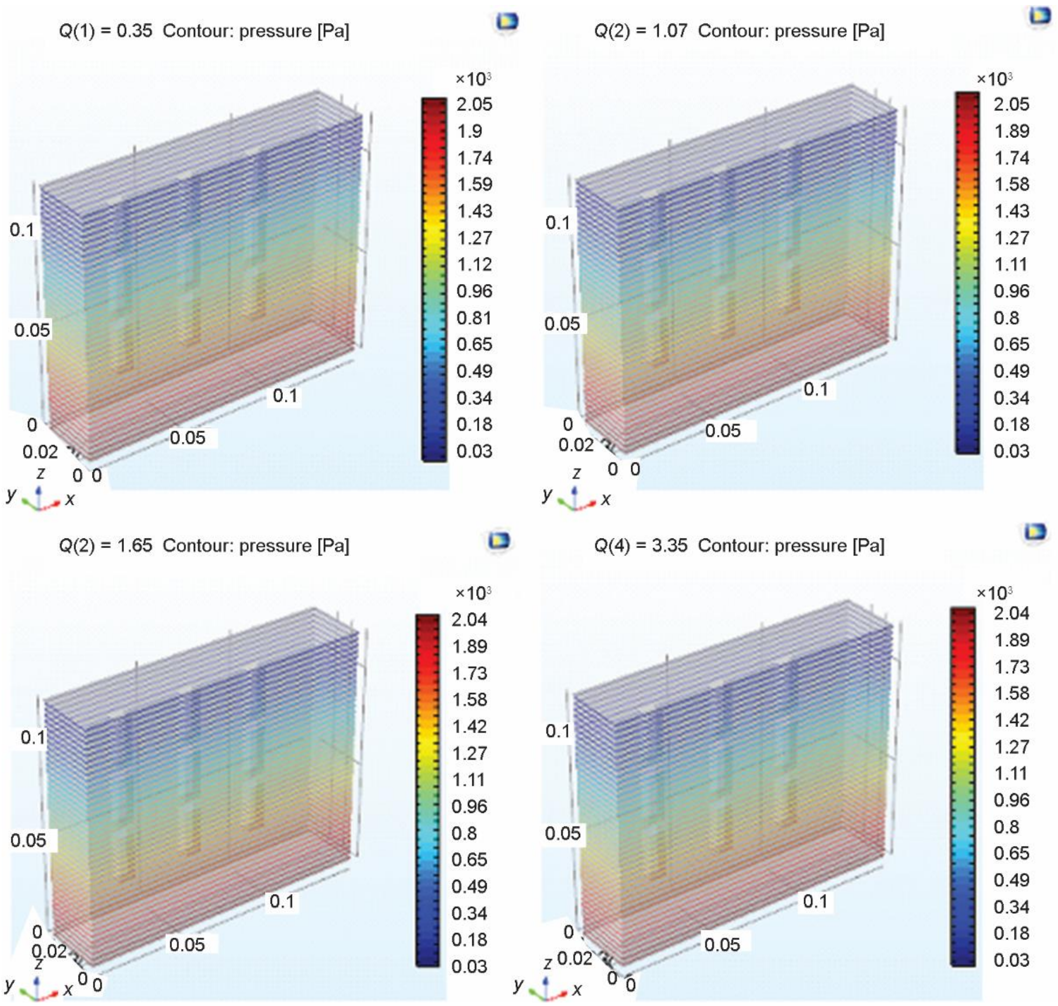

Figure 8. Pressure contours for different power settings (*for color image see journal web site*)

Conclusions

A numerical study of natural convection in a rectangular enclosure filled with a dielectric fluid is analyzed using the COMSOL© commercial software. A 3×3 array of simulated chips is attached to one of the vertical walls, each of which dissipates heat into the fluid by convection. The parameters for heat transfer, the Nusselt number and the Rayleigh number, are calculated and compared with those obtained in a series of previous experiments conducted by the author of this study. There is a maximum difference of 36% between the experiments and the COMSOL© results for the Rayleigh number range of $5.0 \cdot 10^7$ to $3.3 \cdot 10^8$. This difference could result from the assumptions of the CFD package, such as the Boussinesq approach. Also the combined effect of the uncertainties in the experimental results mentioned in the introduction section is another source of deviation of numerical results from the experimental ones. Although there are some correlations in the literature to calculate the natural convection heat transfer in enclosures, these are mostly intended for basic cases, for example, for the case where

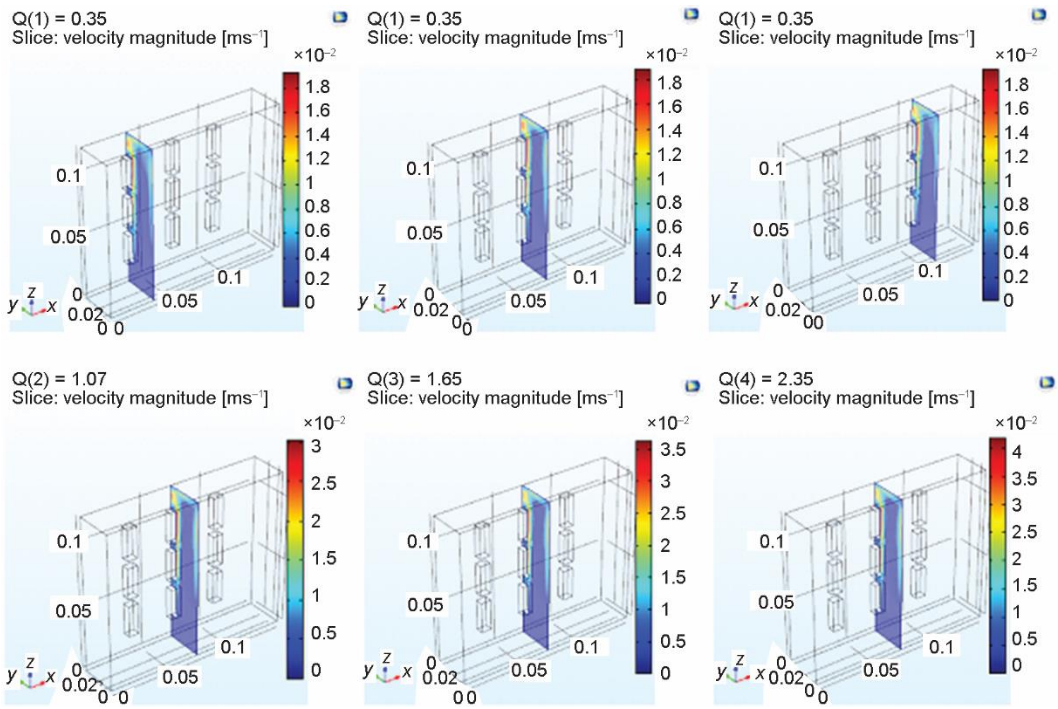

Figure 9. Velocity distribution at middle planes for different power settings
(*for color image see journal web site*)

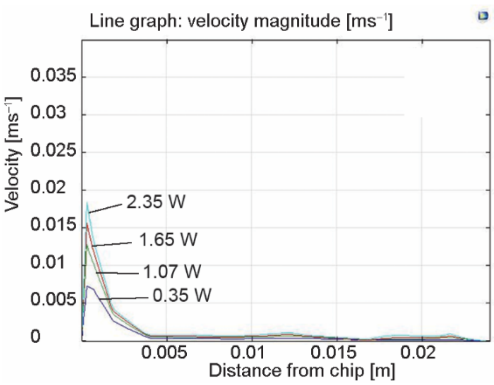

Figure 10. Velocity distributions away from the surface of bottom chip

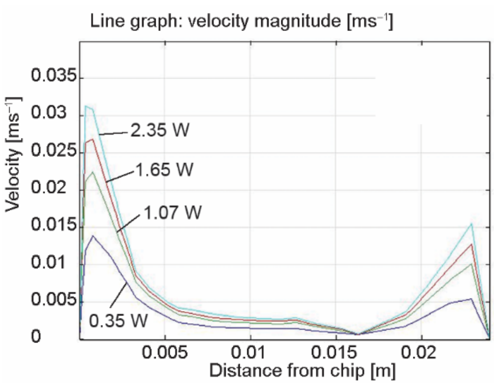

Figure 11. Velocity distributions away from the surface of middle chip

one of the walls is heated while others are cooled or insulated. Enclosures with discrete heaters are not presented as correlations since the number the configurations is unlimited. Numerical methods are essential in certain cases, such as those presented in this study, as long as the results are validated against experimental data. Once the reliability of the numerical model has been validated, various different heat source configurations can be numerically evaluated reliably and without further similar experimentation. It has been shown that COMSOL© is a powerful computational tool in dealing with natural convection in enclosures. Important conclusions similar to those obtained in previous experiments are as follows:

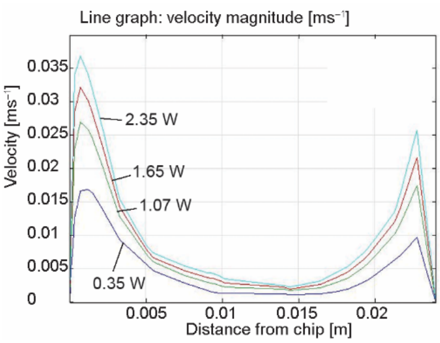

Figure 12. Velocity distributions away from the surface of top chip

- Nusselt number increases with increasing Rayleigh number at a diminishing rate.
- The closer to the top the lower the Nusselt number.
- Irrespective of the column chips at the same row have almost the same Nusselt number values reducing this problem to a quasi-2-D one.

Having successfully validated the simulation results using those of the experiments, it is concluded that numerical model implemented herein can be confidently utilized for similar designs of electronic circuits in confined spaces. As a future work, the numerical model described herein can also be validated by implementing other CFD packages, *e. g.* ANSYS FLUENT© or Openfoam® by using the same model. The deviations in results obtained from these alternative software can be related to the solution methods used in the individual packages.

Nomenclature

C_p – specific heat, [$\mathrm{Jkg^{-1}C^{-1}}$]
$\vec{\mathrm{F}}$ – body force vector, [$\mathrm{Nm^{-3}}$]
Gr – Grashof number, [–]
g – gravitational acceleration (= 9.807), [$\mathrm{ms^{-2}}$]
H – enclosure height, [mm]
h – heat transfer coefficient [$\mathrm{Wm^{-2}K^{-1}}$]
k – conduction coefficient [$\mathrm{Wm^{-1}K^{-1}}$]
L – chip height, [mm]
Nu – Nusselt number, [–]
Pr – Prandtl number, [–]
Q – Joule heating, [–]
q – heat flux at the chip surface, [$\mathrm{Wm^{-2}}$]
Ra – Rayleigh number, [–]
T – temperature [K]
$\vec{\mathrm{u}}$ – velocity vector [$\mathrm{ms^{-1}}$]

Greek Symbols:

β – thermal expansion coefficient, [$\mathrm{K^{-1}}$]
μ – dynamic viscosity (= $\rho\nu$), [Pa.s]
ν – kinematic viscosity [$\mathrm{m^2s^{-1}}$]
ρ – density [$\mathrm{kgm^{-3}}$]

Subscripts

c – cold, denoting heat exchangers
f – film
s – chip surface

References

[1] Faghri, A., *et al*., *Advanced Heat and Mass Transfer*, Global Digital Press, Kursawan, India, 2010, pp. 515-590
[2] Incropera, F. P., Convection Heat Transfer in Electronic Equipment Cooling, *J. Heat Transfer*, *110* (1988), 4b, pp. 1097-1111
[3] Pamuk, M. T., Natural Convection Immersion Cooling of an Array of Simulated Chips in an Enclosure Filled with Dielectric Fluid, M. Sc. thesis, Naval Postgraduate School, Monterey, Cal., USA, 1987
[4] Aytar, E., Natural Convection Immersion Cooling of an Array of Heated Protrusions in an Enclosure Filled with Dielectric Fluid: Effects of Enclosure Width and Fluid Prandtl Number, M. Sc. thesis, Naval Postgraduate School, Monterey, Cal., USA, 1991
[5] Gunes, H., Three-Dimensional Thermal Convection in Enclosures and Vertical Channels: Applications to Cooling of Electronic Equipment. M. Sc. thesis, Lehigh University, Bethlehem, Penn., USA, 1992
[6] Hermes, C. J. L., *et al*., A CFD Model for Buoyancy Driven Flows inside Refrigerated Cabinets and Freezers, *Proceedings*, International Refrigeration and Air Conditioning Conference, Purdue University, West Lafayette, Ind., USA, Paper 608, 2002
[7] Basak, T., *et al*., Effects of Thermal Boundary Conditions on Natural Convection flows within a Square Cavity, *International Journal of Heat and Mass Transfer*, *49* (2006), 23-24, pp. 4525-4535

[8] Turan, O., *et al*., Laminar Natural Convection of Power-Law fluids in a Square Enclosure with Differentially Heated Side Walls Subjected to Constant Temperatures, *Journal of Non-Newtonian Fluid Mechanics*, *166* (2011), 1, pp. 1049-1063
[9] Bazylak, A., *et al*., Natural Convection in an Enclosure with Distributed Heat Sources, *Numerical Heat Transfer. Part A*, *49* (2006), 7, pp. 655-667
[10] Varol, Y., *et al*., Natural Convection in Triangular Enclosures with Protruding Isothermal Heater, *International Journal of Heat and Mass Transfer*, *50* (2007), 13-14, pp. 2451-2462
[11] Doghmi, H., *et al*., Three-Dimensional Mixed Convection Heat Transfer in a Partially Heated Ventilated Cavity, *Thermal Science, 24* (2020), 3B, pp. 1895-1907
[12] Purusothaman, A., *et al*., An Analysis on Free Convection Cooling of a 3×3 Heater Array in Rectangular Enclosure using Cu-EG-Water Nanofluid, *Journal of Applied Fluid Mechanics*, *9* (2016), 6, pp. 3147-3157

Paper submitted: August 16, 2019
Paper revised: October 25, 2019
Paper accepted: November 8, 2019

© 2021 Society of Thermal Engineers of Serbia.
Published by the Vinča Institute of Nuclear Sciences, Belgrade, Serbia.
This is an open access article distributed under the CC BY-NC-ND 4.0 terms and conditions.