ON THE NUMERICAL SIMULATION OF FINS IN NATURAL CONVECTION

De Schampheleire S.*, De Kerpel K., Ameel B., Sharif A., Bagci O. and De Paepe M.
*Author for correspondence
Department of Flow, Heat and Combustion Mechanics,
Ghent University,
9000 Ghent
Belgium,
E-mail: Sven.DeSchampheleire@ugent.be

ABSTRACT

In the numerical study of heat sinks it is known from open literature that a sufficient amount of fluid domain should be added at each side of the heat sink. However, the question in this context is: what can be defined as sufficiently far away from the heat sink? This work studies how the size and location of the fluid domain affects the calculated heat transfer coefficient. The purpose of this study is showing the large uncertainties that are implied by adding an insufficient amount of fluid domain. Three fin row types are studied: a rectangular, an interrupted rectangular and an inverted triangular fin row.

First, the influence of adding fluid domain to the sides of the heat sink is studied. A large decrease of the heat transfer coefficient on both sides and bottom is observed. Next, the influence of adding fluid domain on both the top and the sides is studied. For the rectangular fins the impact on the lumped heat transfer coefficient is +12% compared to the case without any fluid domain added. While for the inverted triangular fin shape no net effect is observed on the lumped heat transfer coefficient.

INTRODUCTION

Electronic components are omnipresent in everyday products e.g. in cars, computers and power converters. It is important to reduce the electrical energy used by the electronic components and limit the chance of electronics failure. The main causes of electronics failure are elevated operating temperatures, over-voltage and moisture [1]. The failures caused by elevated operating temperatures can be avoided by proper cooling of the electronic components. The main failure mechanics caused by elevated operating temperatures are diffusion in the semiconductor materials, creep of the bonding materials and chemical reactions. To keep the temperature level and energy use of these components low, one option is to *use heat sinks in buoyancy-driven heat transfer*. In this heat transfer mode, the air flow is caused by buoyancy forces. A heat sink generally consists of a substrate with fins to increase the heat transferring surface area. In open literature many studies on different fin shapes for heat sinks in natural convection are available. One heat sink in buoyancy-driven heat transfer outperforms another when for the same energy dissipation the flow resistance is lower, in this case the temperature of the heat sink will be lower.

The flow patterns induced by heating in natural convection can be understood intuitively. Hot fluid generally has a lower density and therefore rises (flow against gravity) while cold fluid with higher density tends to move downwards (flow with gravity). Heat sinks that operate in buoyancy-driven heat transfer can generally only be used at relatively low heat transfer rates as the substrate temperature typically has to be kept below 70°C. For high heat transfer rates heat sinks in forced convection can be used. The fin shapes for the latter type of heat sink are more complex to manufacture e.g.: the bonded finned, the single finned and/or the skived finned heat sinks [2]. Hence, heat sinks made to operate in natural convection imply a lower investment, maintenance and operating costs.

NOMENCLATURE

H	$[m]$	fin height
L	$[m]$	fin length
S	$[m]$	fin spacing
T	$[K]$	temperature
Q	$[W]$	heat transfer rate

Greek symbols

β	$[-]$	thermal expansion coefficient

Subscript

avg	average
env	environment
hot	hot surface
tot	total

Work done on buoyancy-driven convection

Most research found in open literature on this topic is quite diffuse, due to the many parameters involved in buoyancy-driven heat transfer:

- Characterization of the *heat sink geometry*. The heat sink material has to be reported. Another important parameter is the thickness of the fin itself, which is not always reported [3].
- Characterization of the *test rig*. In order for the experiments to be repeatable, the test rig has to be described properly. A discussion on this topic can be found in De Schampheleire et al. [4].

- *Emissivity* of the material. Most of the time, the authors are neglecting radiation.

However, Sahray et al. [5] found that for dense pin fin heat sinks the radiative contributions were up to 45% of the total heat transfer rate. The same was observed by Sparrow and Vemuri [6] in their study on the orientation and radiation in pin fins. They found that fractional contributions of radiation to the combined heat transfer were generally in the 25%-40% range with larger contributions for the smaller temperature differences between substrate and environment.

Ideally, the fin material has to be polished to reach an emissivity value around 0%. This is the only possible way to compare correlations with experimental results or other correlations. However, this is practically not feasible. Instead one should measure the emissivity and report correlations without the radiative influence.

- *Inclination angle*. The impact of the inclination angle is very depended on the fin shape [7].
- *Enclosure*. The enclosure in which the heat sink is placed of course has a big impact on the performance [4].

All these dependencies make it quite difficult to optimize and test one specific heat sink for all these parameters. To limit the amount of necessary experiments, some work is also done on numerical simulation of heat sinks in natural heat transfer with the help of computational fluid dynamics (CFD). As this is the topic of current paper, the methodology for CFD calculations is discussed further in the following section.

CFD simulations

Ahmadi et al. [8] studied natural heat transfer in vertically-mounted rectangular interrupted fins. The authors used a **2D numerical model** and included **radiation**. Most of the authors do include radiative influence, however authors like Dialameh et al. [9] neglected it without given a profound reason for this hypothesis. On the other hand, it is quite rare to study natural convection with a 2D numerical model (like Ahmadi et al.). All other authors referred in this work use 3D.

The fluid in Ref. [8] is modelled through the Boussinesq approximation. Although this approximation is frequently used in literature, it can only be used when the changes in actual density are small, specifically it is only valid when $\beta(T - T_0) \ll 1$. In case of Ahmadi's work, $\beta(T - T_0)$ varies between 0.15 and 0.24. Questions can be raised whether $\beta(T - T_0) \ll 1$ is satisfied. Instead of the Boussinesq approximation some authors use air incompressible ideal gas [10] or ideal gas [7] or constant properties expect for a temperature dependent density [3]. **Turbulence models** are most of the time not used, expect for the work published by Tari et al. [7], where they studied the complete heat sink surrounded with 3 m on 3 m fluid domain. They used a zero order turbulence model, which will limit the accuracy of the results because of the addition of numerical diffusion in the system.

Mesh and discretization

Most authors only simulate one or half of a fin row [8-10], except Tari et al. [7] who studied the complete heat sink. However there are larger differences when looking to the fluid domain that the authors have (not) added to the heat sink. Ahmadi et al. [8] have added only a fluid domain in front of the simulated fin, but not after the fin's end. Shen et al. [10] added fluid domain on all sides of the fin and in contrast Dogan et al. [3], who studied several types of fins without adding any fluid domain.

Next, there is also a large deviation over the grid discretization that is used. First of all, some authors do not report their grid, like Tari et al. [7]. Ahmadi et al. [8] mention an 'optimum' grid size of 1 mm, while Shen et al. [10] mention 2 mm and Dogan et al. [3] 0.5 mm as smallest grid size used.

Purpose of this study

The authors will propose a method of how to study heat sinks in buoyancy-driven convection. Our base case of the work of Dogan et al. [3] (2014). Dogan et al. have studied different fin shapes (from rectangulars to inverted triangles). However, no fluid domain around the heat sink was added. The

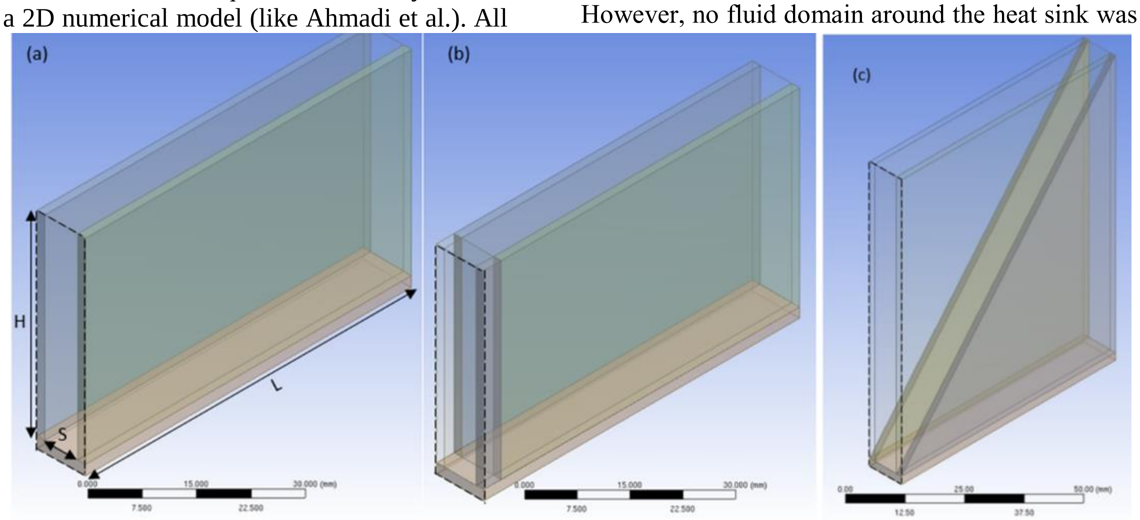

Figure 1. Illustration of the used fin shapes in this work. (a) rectangular fin, (b) interrupted rectangular fin, (c) inverted triangular shape. Dashed line indicates the symmetry plane.

effect of adding fluid domain on the heat transfer coefficient(s) will be discussed in this work. Furthermore, an uncertainty analysis will be presented for the numerical results. In this way, one is able to determine how fine the grid discretization has to be for this study.

NUMERICAL CALCULATIONS

Used geometry

In this work, the considered fin shapes are (see Figure 1):
(a) A rectangular fin shape (Fig. 1(a)) with a length (L) of 127 mm, a fin spacing (S) of 6.4 mm and a height (H) of 38 mm.
(b) An interrupted fin shape (Fig. 1(b)), all characteristics are the same as those for the rectangular fin shape, except that in the middle of the fin 7 mm is cut away. It seems valuable to study this interrupted fin design since the simulations by Dogan et al. [3] show that the temperature in the middle of this fin was approximately the substrate temperature.
Figure 1 (c) shows an inverted triangular fin shape with a length (L) of 127 mm, a fin spacing (S) of 6.4 mm and a height (H) of 76 mm. For all cases, a fin thickness of 3 mm was used. The thickness of the substrate was taken to be 2 mm. For all of the tested fin shapes, 8 different fluid domains were generated. Each of the used geometries got a label. 'Mesh-ref' means that no fluid domain is added. 'Mesh-5mm/10mm/20mm' means that resp. 5-, 10- or 20mm-long fluid domains are added to the sides. 'Mesh-top-10mm/20mm/30mm' means that resp. 10, 20 and 30 mm is added to sides *and* the top. The last two meshes: 'Mesh-top-50mm/100mm' mean that resp. 50 mm and 100 mm at the top, while only adding 20 mm at the sides. This is to minimize the computational time. Also larger domains are tested without a significant impact on the thermal performance. In Figure 2, an illustration is made of the fin surfaces that will be discussed in this work. The illustration is made for the rectangular fin shape. However, for the other fin designs, similar illustrations can be made.

The following heat transfer coefficients are studied (See Figure 2 for the names of the discussed surfaces):

- h_{sides}. This is the heat transfer coefficient calculated as an area-averaged value on the internal sides of the fin row (left and right).
- h_{bottom}. This is the heat transfer coefficient calculated as an area-averaged value on the bottom side of the fin row.
- $h_{with\ radiation} = \frac{Q_{tot}}{A_{fins} \cdot (T_{hot} - T_{env})}$. This heat transfer coefficient is an average heat transfer coefficient over all the available fin surfaces. If a fluid domain is added to e.g. the sides of the fin row, the heat transfer coefficient of the frontal fins will also be taken into account.
- $h_{with\ radiation,lim}$. This is the heat transfer coefficient only for the left, right and bottom side of the fin, independent of which mesh being studied. This heat transfer coefficient is an area-averaged value of h_{sides} and h_{bottom}.

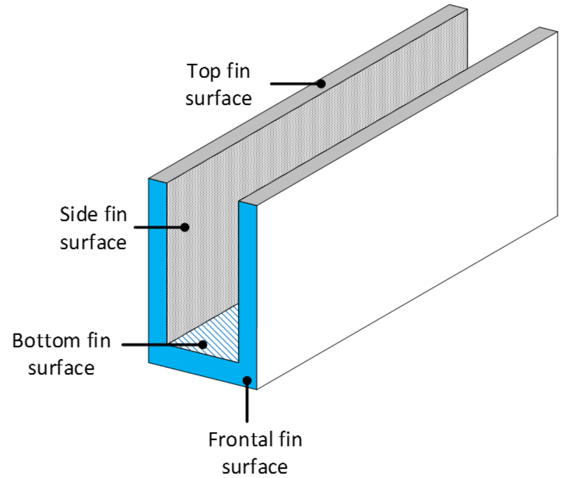

Figure 2. Illustration of the discussed fin surfaces.

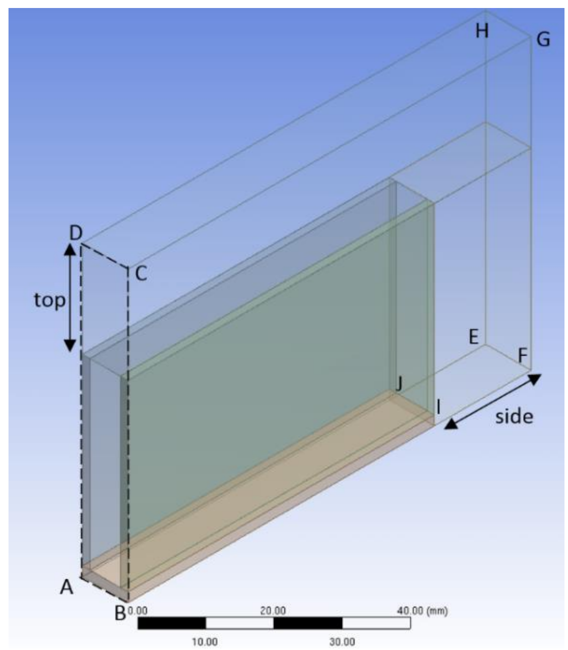

Figure 3. Illustration of the boundary conditions in case of a rectangular fin row ('Mesh-top20mm')

Governing equations and boundary conditions

In addition to the standard set of Navier-Stokes equations, the radiative heat transfer is taken into account. In this work, the surface to surface model for radiation in the CFD package by Ansys® was used. An emissivity value of 1 is assumed for sake of simplicity. Furthermore, the pressure-velocity coupling in this work is done with the SIMPLE-algorithm, while momentum and energy is calculated via second order upwind discretization. For each cell the mass (Eq. (1)), momentum (Eq. (2)) and energy equation (Eq. (3)) for the fluid are solved steady together with the energy conduction equation for the solid (Eq. (4)). For the momentum term this is the so-called incompressible Navier-Stokes equation in convective form.

$$\nabla \cdot (\rho \vec{v}) = 0 \quad (1)$$

$$(\vec{v} \cdot \nabla)\vec{v} - \nu \nabla^2 \vec{v} = -\frac{1}{\rho_0} \nabla p + \vec{g} \quad (2)$$

$$\vec{v} \nabla (\rho_0 T) = \frac{\mu}{Pr} \nabla^2 T \quad (3)$$

$$\nabla^2 T = 0 \quad (4)$$

Air is modelled as an incompressible ideal gas. Therefore the operating density (ρ_0) has to be given. This operating density is taken at ambient temperature (20°C).

In Figure 3, the applied boundary conditions are illustrated for a rectangular fin row. For the two other fin shapes considered, the boundary conditions were exactly the same. Symmetrical boundary conditions are used for surfaces ABCD, BCGF and ADHE. The surface EFIJ is assumed to be adiabatic. This is not correct when compared to practical applications, however, in this work the focus lays on comparing different fin shapes. ABJI is the heated surface. Unless otherwise stated, the boundary condition at the heated surface is a fixed flux of 2250 W/m², in all the studied cases. This is exactly the same boundary condition as used by Dogan et al. [3]. Surface HGEF is a so-called free surface where the gauge total pressure is set to 0 Pa. For surface DCHG the gauge static pressure is fixed at 0 Pa.

The discretization is done exactly the same for the three studied fin shapes: thickness of the fin materials is 0.09 mm/cell, fin height and length and adjacent fluid domain is 0.24 mm/cell and width of the fluid in between the fins was 0.08 mm/cell for our study. The finest discretization used by Dogan et al. [3] was 0.5 mm/cell.

Similar to Dogan et al. [3], other authors like Dialameh et al. [9] and Shen et al. [10] also compare only several grid discretization that are very close to each other. For example Dialameh et al. [9] have tested 20x40x45, 19x50x55 and 24x50x60 as a discretization scheme. They observed no impact on the average heat transfer coefficient, so they assumed the coarsest grid to be acceptable. However, in order to estimate an uncertainty level of the grid discretization compared to an infinitely fine grid, one has to refine the grid preferably with a factor 2 in each direction [11]. Or at least with a constant growth factor, about 1.1, for example. However, none of the mentioned authors performed such a procedure, which raised doubt on the viability of the computational results. In this work, the Richardson extrapolation is used to obtain a higher order estimate of the continuum value [11]. This continuum value is the value that would be obtained for an infinitely fine grid. The uncertainty analysis is only performed on the rectangular fin case without fluid domain, however, similar results hold for the other fin shapes. The grid discretization as explained above is coarsened with a factor 2 in each direction in order to perform the uncertainty analysis. This grid is called the coarse grid in Table 1. The results for the average heat transfer coefficient for different fin faces for both studied grids are shown in Table 1. The uncertainties are calculated according to Roache [11]. The largest relative uncertainty on the heat transfer coefficient is found on the right and left side of the fin. When calculating with the finest grid the relative uncertainty is 1.8%, while for the coarse grid this is 7.1%. Also note that the coarse grid tested in this work is already much finer than the used grid in the work of Dogan et al. [3] where the uncertainties will be significantly higher than the results for our coarse grid.

	Coarse grid	**Fine grid**
h_{sides} $[W/m^2K]$	3.394	3.455
$h_{bottom}[W/m^2K]$	4.724	4.801
$h_{ave}[W/m^2K]$	4.621	4.697
$E_{sides}[\%]$	**7.07**	**1.77**

Table 1. Uncertainty analysis for grid discretization

RESULTS AND DISCUSSION

In Figures 4 to 7 the heat transfer coefficients for the different studied meshes are reported. First the addition of only a fluid domain at the sides will be studied (first section). In the second section, the effect of adding fluid at sides and on top will be investigated.

Influence of adding fluid to the sides

Adding fluid domain at the sides of the fin material has a large impact on h_{bottom} and h_{sides} as can be seen from Figures 4 and 5. The heat transfer coefficient on both left, right and bottom side of the fin row decreases by adding extra fluid domain. The heat transfer coefficient h_{sides} decreases by 18% for the rectangular and interrupted fins. The impact for the inverted triangular fin is approximately the same: a decrease of around 20%. For the heat transfer coefficient at the bottom side of the fin row, h_{bottom}, the impact is even larger: a decrease of around 28% is observed, independent of the type of fin row that is simulated.

Looking to the resulting velocity profile at the inlet of the fin row all velocity vectors will be perpendicular to the inlet plane, as was also the case in Dogan et al. [3]. However, with fluid domain around the side of the heat sink, the vector of the fluid velocity is not perpendicular to its inlet plane anymore. Therefore, the fluid enters the inlet plane at an angle, as can be seen by the streamlines shown in Figure 8. This angle is negative for the upper half of the fin row, while it is positive for the lower half referred to the orientation of the substrate. The fact that the lower half has a positive angle is caused by the substrate: the incoming fluid heats up along the frontal fins, just before entering the fin row. This also explains why the impact on the heat transfer coefficient at the bottom side of the fin row (h_{bottom}) is, in all cases, larger compared to the impact on the heat transfer coefficient at the sides (h_{sides}).

The combined impact on $h_{with\ radiation,lim}$, an area average of h_{bottom} and h_{sides}, is thus strongly negative and independent of the kind of fin row as is derived from Figure 7. However, the impact on the heat transfer coefficient $h_{with\ radiation}$ is much more modest (see Figure 6). This is due to the compensation effect of the frontal fins, just before the entrance of the fin row. The absolute values for the heat transfer coefficient $h_{frontal}$ are much larger than the values for h_{bottom} and h_{sides} (not reported here).

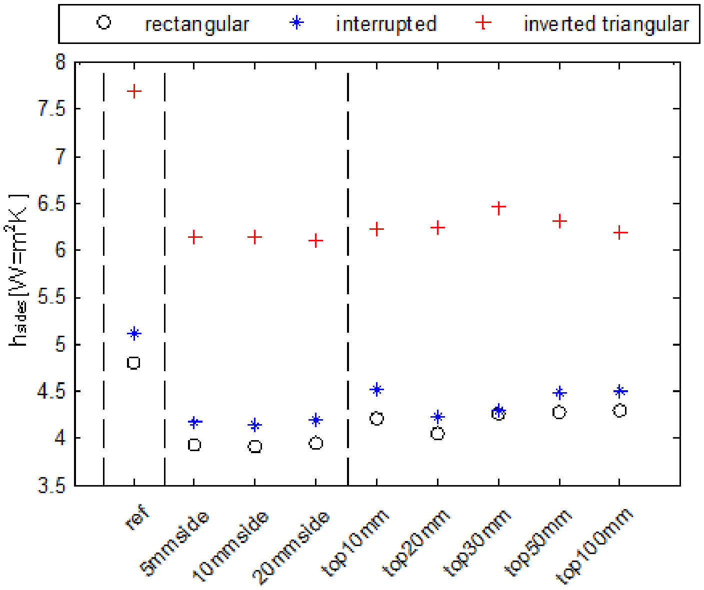

Figure 4. Dependency of the heat transfer coefficient on the fluid domain at the sides of the fin row (h_{sides})

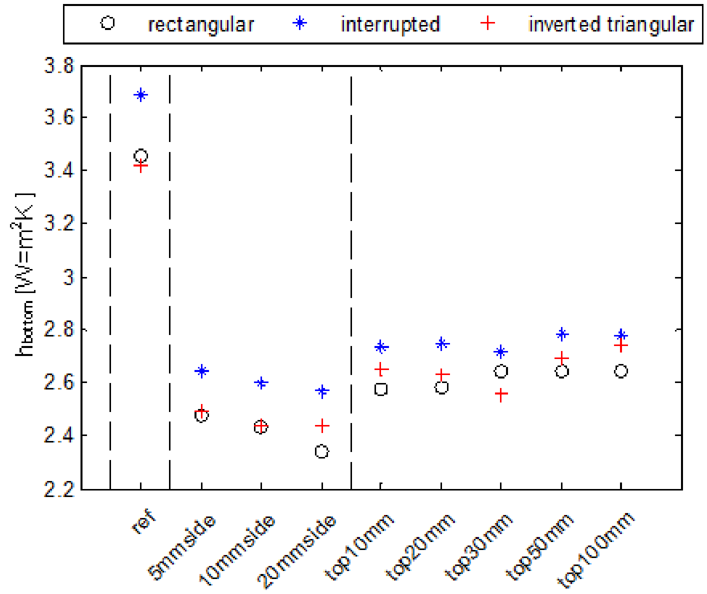

Figure 5. Dependency of the heat transfer coefficient on the fluid domain on the bottom side of the fin row (h_{bottom}).

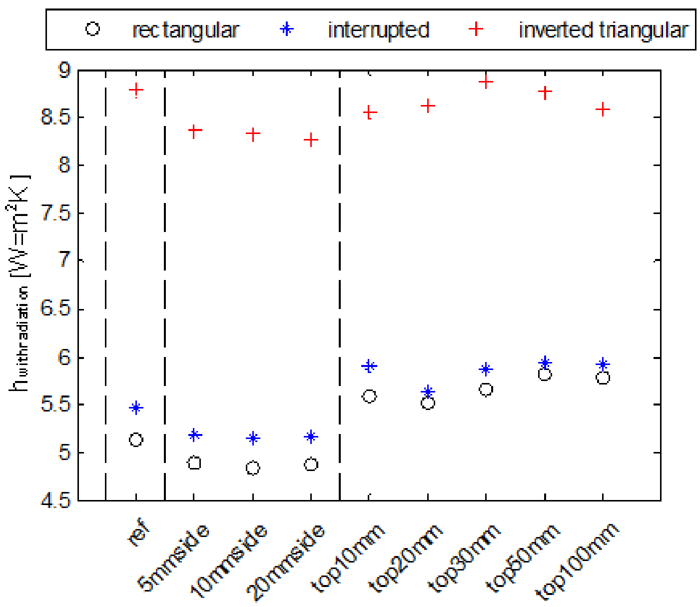

Figure 6. Dependency of the heat transfer coefficient on the fluid domain, taken all the available surfaces into account ($h_{with\ radiation}$).

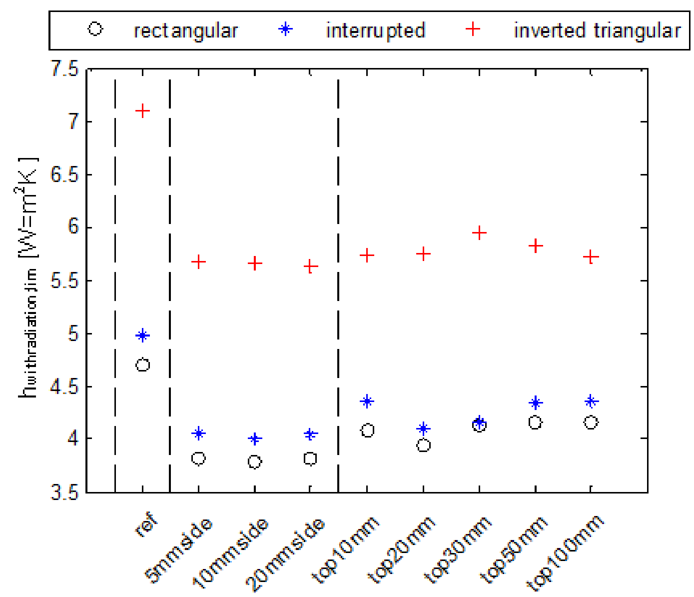

Figure 7. Dependency of the heat transfer coefficient on the fluid domain, taking an area-average of h_{sides} and h_{bottom} into account ($h_{with\ radiation,lim}$).

As can be seen from Figures 4 to 7, the difference in heat transfer coefficient of adding 5 mm, 10 mm or 20 mm to the sides of the fin row has a negligible effect, taking into account the relative uncertainty of 2% on the numerical results. In other words, there is no need of adding a large amount of fluid domain, in fact for this case 5 mm of fluid domain was enough. However, this is only the case when the effect of adding fluid domain to the top of the fin row is neglected. As will be shown in next section, where both a fluid domain at the sides *and* at the top will be added, the flow pattern will completely change again and there will be a combined effect of adding fluid domain at the sides and flow domain at the top.

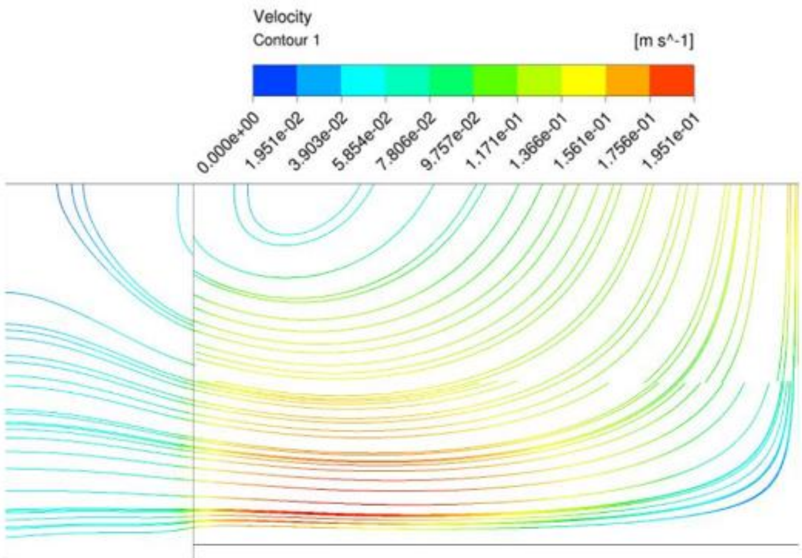

Figure 8. Velocity contour plot and streamlines in the middle of the rectangular fin row with 20 mm of fluid domain added at the sides ('Mesh-20mmside', right side: symmetry plane)

Influence of adding fluid to the top (and sides)

Adding fluid domain to the top and both sides of the fin row will change the heat transfer coefficients further. Figure 5 shows the effect of adding fluid domain to top and sides on h_{bottom}. Although the impact on the heat transfer coefficient is still large and negative compared to the reference geometry ('Mesh-ref'), the relative impact is smaller compared to the case where only fluid domain was added to the sides. For the rectangular and interrupted fin design the impact on h_{bottom} decreases from -28% to -24%. For the inverted triangular fin the impact on h_{bottom} decreases from -27% to -20% (see Figure 5). The fact that h_{bottom} decreases by adding fluid domain at the sides is explained in previous section. The fact that the impact of adding fluid domain on the top and the side is more severe for the inverted triangular shape can be explained by comparing Figure 9 (for the inverted triangular fin) with Figure 10 (for the rectangular fin). Figure 10 shows that e.g. the streamline in the middle of the fin are oriented more to the bottom side of the fin row compared to the streamlines at the same location for inverted triangular case (Figure 9). This explains the differences in impact for h_{bottom} between the rectangular and inverted triangular fins when adding fluid to top and sides.

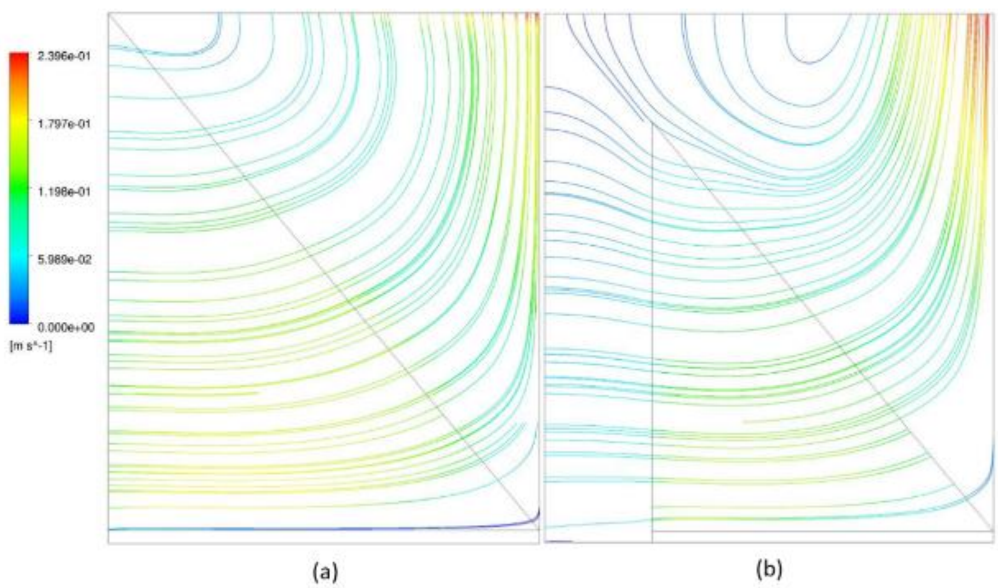

Figure 9. Velocity contour plot and streamlines in the middle of the inverted triangular fin row, (a) without added fluid domain ('Mesh-ref'), (b) with 20 mm of fluid domain added at the sides and top ('Mesh-top20mm', right side: symmetry plane).

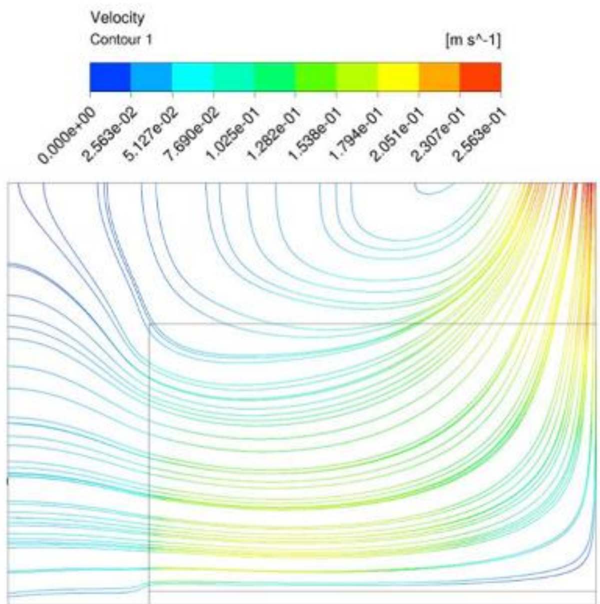

Figure 10. Velocity contour plot and streamlines in the middle of the rectangular fin row with 20 mm of fluid domain added at the sides and top ('Mesh-top20mm', right side: symmetry plane)

Comparing the streamlines in both Figure 9 (inverted triangular case) and Figure 10 (rectangular case), it is clear that the impact of adding extra fluid domain on top and sides of the fin row on h_{sides} is larger for the rectangular and interrupted fin row than for the inverted triangular fin. For the first two fin rows, the impact on h_{sides} is reduced from -18% to -11%, while for the inverted triangular fin the impact on h_{sides} is decreased (not significantly) from -20% to -19% (see Figure 4). This can be explained by the fact that the heated area in case of the inverted triangular fin row is varied in the axial direction. Compared to the rectangular case the heated area decreases in the axial direction of the fin. This means that a higher outflow is induced compared to the rectangular case, causing the lasting negative impact for the inverted triangular case.

Next, the influence of adding extra fluid domain: going from 10 mm to 100 mm is significant in case of h_{sides} and $h_{with\ radiation}$ as can be seen in Figure 4 and 6. The impact on $h_{frontal}$ of adding extra fluid domain on the top is limited (not reported here).

These results show that the conclusions from e.g. the work of Dogan et al. [3] could significantly differ from the correct values due to the lack of fluid domain around the fin row. As shown in this work, a fluid domain has to be added around the fin row. The impact of the fluid domain depends on the studied fin type too.

CONCLUSIONS

The influence of only adding fluid domain to the sides is studied separately. A large decrease is observed on the heat transfer coefficients on both the sides and the bottom of the fin row. This is caused by the fact that by adding fluid domain, the velocity vector is not perpendicular to the inlet of the fin row, inducing a different heat transfer coefficient. Furthermore, the lumped heat transfer coefficient is also studied. This is the average heat transfer coefficient over all the fin surfaces. Adding fluid domain to the sides, also causes a decrease in this lumped heat transfer coefficient. However, the decrease in this lumped value is much more modest, because of the importance of the heat transfer coefficient at the top faces of the fin row.

Finally, the influence of adding fluid domain on both top and sides is studied. Here, the effect of the fluid domain on the heat transfer coefficient is only significant for the rectangular and interrupted rectangular fin shape. Adding fluid domain to all sides of the fin row induces a relative increase in lumped heat transfer coefficient of +12%. However, for the inverted triangular shape, the lumped heat transfer coefficient doesn't change significantly. This means that the conclusions of studying different fin geometries with and without fluid domain can be completely different. Furthermore, a fluid domain should always be added in order to compare different fin structures properly.

REFERENCES

[1] Burris M., How components fail and how to identify failed components, http://components.about.com/od/Components/a/Component-Failures.htm, last accessed: 4 November 2015

[2] Nagarani N., Mayilsamy K., Murugesan A., Kumar G.S., Review of utilization of extended surfaces in heat transfer problems, *Renewable and Sustainable Energy Reviews*, Vol. 29, 2014, pp. 604-613.

[3] Dogan M., Sivrioglu M., Yılmaz O., Numerical analysis of natural convection and radiation heat transfer from various shaped thin fin-arrays placed on a horizontal plate-a conjugate analysis, *Energy Conversion and Management*, Vol. 77, 2014, pp. 78-88.

[4] De Schampheleire S., De Jaeger P., Reynders R., De Kerpel K., Ameel B., T'Joen C., Huisseune H., Lecompte S., De Paepe

M., Experimental study of buoyancy-driven flow in open-cell aluminium foam heat sinks, *Applied Thermal Engineering*, Vol. 59, 2013, pp. 30-40.

[5] Sahray D., Shmueli H., Ziskind G., Letan R., Study and Optimization of Horizontal-Base Pin-Fin Heat Sinks in Natural Convection and Radiation, *Journal of Heat Transfer*, Vol. 132, 2009, pp. 012503-012503.

[6] Sparrow E.M., Vemuri S.B., Natural Convection/Radiation Heat Transfer From Highly Populated Pin Fin Arrays, *Journal of Heat Transfer*, Vol. 107, 1985, pp. 190-197.

[7] Tari I., Mehrtash M., Natural convection heat transfer from horizontal and slightly inclined plate-fin heat sinks, *Applied Thermal Engineering*, Vol. 61, 2013, pp. 728-736.

[8] Ahmadi M., Mostafavi G., Bahrami M., Natural convection from rectangular interrupted fins, *International Journal of Thermal Sciences*, Vol. 82, 2014, pp. 62-71.

[9] Dialameh L., Yaghoubi M., Abouali O., Natural convection from an array of horizontal rectangular thick fins with short length, *Applied Thermal Engineering*, Vol. 28, 2008, pp. 2371-2379.

[10] Shen Q., Sun D., Xu Y., Jin T., Zhao X., Orientation effects on natural convection heat dissipation of rectangular fin heat sinks mounted on LEDs, *International Journal of Heat and Mass Transfer*, Vol. 75, 2014, pp. 462-469.

[11] Roache P.J., QUANTIFICATION OF UNCERTAINTY IN COMPUTATIONAL FLUID DYNAMICS, *Annual Review of Fluid Mechanics*, Vol. 29, 1997, pp. 123-160.