XLI CILAMCE
XLI Ibero-Latin American Congress on Computational Methods in Engineering
16-19 November, 2020 FOZ DO IGUAÇU PARANÁ - BRAZIL

Effect of Thermal Radiation on Natural Convection Cooling in a Protruding Heater Channel

Victor H. Sarmento, André D. Rocha

Centre for Engineering, Modeling and Applied Social Sciences, Federal University of ABC
Av. Dos Estados 5001, 09210-580, Santo André/SP, Brazil
v.hassun@aluno.ufabc.edu.br, a.damiani@ufabc.edu.br

Abstract. Electronic devices are becoming smaller every time and also more powerful, which leads to greater amount of heat that needs to be dissipated. Cooling systems by natural convection do not need power and do not involve movable parts to work, therefore they are simpler and cheaper to build and maintain. This paper investigates numerically the effects of radiation on a cooling system by natural convection. The problem is assumed as bidimensional and the geometry is formed by two parallel plates and there is a protruding heater in one of them, while the other one is insulated. The heat flux from the module is responsible by driving the flow inside the channel. For laminar natural convection, effects of radiation on the cooling were investigated. At first, neglecting thermal radiation, the numerical results, were validated with the literature. After validation, thermal radiation was taking account and the effects of emissivity and width channel on Nusselt number were computed and compared with those without radiation.

Keywords: thermal radiation, natural convection, channel flow, computational fluid dynamics

1 Introduction

The natural convection cooling system holds its advantage from the fact that no additional power is necessary to cause convection and also no moving parts are needed which might make the whole system cheaper to build and maintain. In addition, the system is quite silent and trustful, being widely used in systems that do not need to dissipate huge amounts of heat. Natural convection cooling systems have been extensively studied by numerically or experimentally means. Rocha [1] investigated experimentally an array of 7 equally powered protruding elements that were disposed in vertical plate equally spaced, he managed to relate the Nusselt number to the Rayleigh number. Earlier, Kang and Jaluria [2] also performed a experimental study on natural convection for both horizontal and vertical plates, stating that the temperature rises fast as the Grashof number increases and this increase is larger for wall plumes (vertical plates). A strong dependence on the module aspect ratio was also found by them. Desrayud and Fichera [3] studied numerically the natural convection cooling in a channel with a protruding heat-flux module and the effects of module dimension, position, Rayleigh number and channel aspect ratio into a correlation to find out the module temperature. Avelar and Ganzarolli [4] analyzed natural convection in channels with protruding heater modules both experimental and numerically with good agreement between the results. Hung and Shiau [5] investigated experimentally natural convection heat transfer in a channel formed by parallel plates claiming that channel spacing has no significant effect on heat transfer characteristics while the heat transfer performance is strictly to heat-flux. Also, the recirculation zones have lower heat transfer coefficients. Said and Krane [6] studied by experimental and numerical means laminar natural convection flow inside an obstructed channel. Experimental and numerical results were in close agreement. They discussed the effects of obstruction position, stating that as the obstruction gets closer the channel's exit, the average heat transfer for the channel is reduced.

In most of the previous studies on natural convection in channels with protruding heaters, the influence of radiation is neglected. There has been little study on the coupled heat transfer problem involving both convection and radiation. However, it is well known that when natural convection in air is involved, the heat transfer by convection and radiation are usually of the same order of magnitude. Sarper et al. [7] investigated both numerically and experimentally the effect of the surface radiation in a natural convection laminar driven flow in vertical channel and they observed significant changes on the unheated wall temperature and also the disappearance of reverse flow.

Tkachenko et al. [8] also investigated the radiation effects on natural convection in channel flows, however, they considered the flow as turbulent and observed, as expected, lower temperatures, therefore, higher heat transfer rates when radiation is taken into account. Andreozzi and Manca [9] investigated the radiation effects on natural convection in vertical channels as well, although they did it experimentally, and noticed 18% temperature decrease when using a plate with lower emissivity. Although these works investigated the radiation effect on natural convection in vertical channels, their studies did not considered a protruding heater module.

The aim of the present work is to study the radiation-natural convection heat transfer in a channel with a single protruding heater module in a laminar flow condition for three Rayleigh numbers and different heater emissivity values.

2 Physical Problem and Numerical Simulation

2.1 Physical Problem

The problem consists of two vertical parallel plates containing one protruding heater module mounted in one plate and at half channel height as shown in Fig. 1. The module is made of aluminum and the plates are made of fiberglass, in order to behave as insulated. Three aspect ratios for the channel were simulated: $A = 5$, $A = 10$ and $A = 12.5$. Three Rayleigh numbers were considered: $Ra = 10^6$, $Ra = 10^5$ and $Ra = 10^4$, therefore, the flow was modeled as laminar. The heat is added to the system at the module base, as demonstrated on Fig. 1, and it dissipates through the protruding element by conduction, later being dissipated by natural convection and radiation.

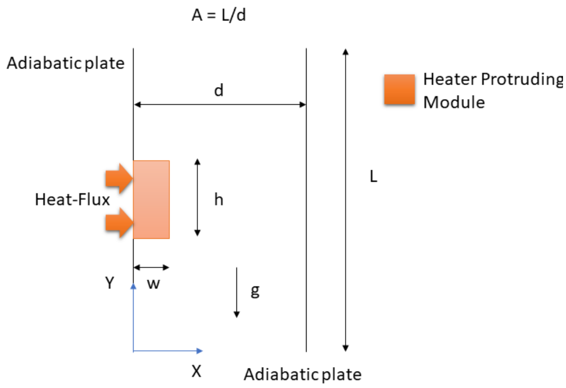

Figure 1. Computational domain. Adapted from Desrayud and Fichera [3]

2.2 Governing Equations

The flow is considered as laminar, bi-dimensional and in steady state. Moreover, the Boussinesq's hypothesis is considered due to its widely application for natural convection driven flows without big temperature gradients. The governing equations for the problem are continuity, momentum and energy equation and they are modified due to some assumptions and shown in eqs. (1, 2, 3 and 4).

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{1}$$

$$\rho(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}) = -\frac{\partial P}{\partial x} + \mu(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}) \tag{2}$$

$$\rho(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y}) = -\frac{\partial P}{\partial y} + \rho_0\beta g(T - T_0) + \mu(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}) \tag{3}$$

$$\rho_0 cp[u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y}] = k_a[\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}] + q_v \tag{4}$$

Where u and v are the velocity in the X and Y direction respectively, P is the static pressure, ρ, μ, β, k_a and cp are density, dynamic viscosity, thermal expansion coefficient, thermal condutivy relatively to the air and heat capacity respectively and q_v is the heat added to the system per volume unit.

$$q_{out,k} = \varepsilon_k \sigma T_k^4 + (1-\varepsilon)_k q_{in,k}. \tag{5}$$

The S2S model equation is shown in eq. 5 representing the energy reflected from the surface k where $q_{out,k}$ is the energy flux leaving the surface, ε_k is the emissivity, σ is Boltzmann's constant and $q_{in,k}$ is the energy flux that reaches the surface from the surroundings.

2.3 Computational Domain and Boundary Conditions

The computational domain is shown in Fig. 1 and it includes the area between the plates. The whole CFD process (pre-processing, processing and post-processing) was done using the commercial software package from ANSYS®, with the Fluent® solver. The channel entrance at the bottom was set as pressure inlet with $P = 1atm$, the same pressure for the channel exit, but it was set as pressure outlet. All the solid-fluid interfaces have the no-slip condition applied and the ambient temperature $T_\infty = 300K$. The heat-flux is added at the base of the protruding heater module and the amount of heat is governed by the eq. (6) like Desrayud [3] defined. The fluid properties correspond to air at $T = 300K$ and are shown in the Tab. 1.

$$Ra = \frac{g\beta q_s" d^4}{k_a v\alpha}. \tag{6}$$

Where α is the air's thermal diffusivity.

Table 1. Fluid properties at 300 K

Property	Value	Unity
k_a	0.0263	[W/mK]
ρ	1.161	[kg/m^3]
μ	1.85E-5	[Pa.s]
α	2.25E-5	[m^2/s]
Pr	0.7062	[-]

The total heat transfer rate by convection q_{conv} between the heater module and the fluid is defined as eq. (7) according to Incropera [10]:

$$q_{conv} = (T_s - T_\infty)\int_{A_s} h dA_s. \tag{7}$$

The Nusselt number (Nu), which measures the convectice heat transfer coefficient, is defined by eq. (8) as stated by Incropera [10]:

$$Nu = \frac{hL}{k_a}. \tag{8}$$

As the plates are insulated, the heat added through the heater module base q is dissipated by convection and radiation as states eq. (9):

$$q = q_{conv} + q_{rad}. \tag{9}$$

Aluminum's emissivity may vary from 0.09 and up to 0.8 for anodized aluminum, therefore the following values were applid in this work: 0.09, 0.15, 0.3, 0.5, 0.6 and 0.8. For the plates, since they represent fiberglass, their emissivity was set as 0.75.

2.4 Numerical Method

The governing equations are solved using the ANSYS® Fluent 19.0 software. Fluent is a Finite Volume Method (FVM) based solver. The algorithm chosen was the pressure-based coupled. It solves simultaneously the continuity and the momentum equation, offering some advantages for convergence when compared to segregated algorithms. The spatial discretization chosen for energy and momentum was the second order upwind, owing to its high discretization and for pressure PRESTO has been chosen, as recommended. Regarding the convergence criteria, the residuals were set to 10^6.

2.5 Discretization Error

The discretization error, intrinsic to CFD applications, was evaluated using the Grid Convergence Index method described by Celik et al. [11] and the fine grid convergence index was 0.08477% More details are available on Tab. (2). Later, the grid used for all the simulations was the intermediate one.

Table 2. Grid Coefficient Index procedure for A = 5 and Ra = 10^6

Mesh type	Cells	Nu	Grid refinement factor (r)	GCI
Coarse	1617	11.68097	-	-
Intermediate	6468	11.60811	2	-
Fine	25872	11.58779	2	0.3034

2.6 Validation

In numerical models, it is important not only to minimize the discretization error, using a method like GCI, but also verifying the accuracy. In order to verify, the simulation has been run without radiation and two major comparisons were made between the results obtained without radiation and the ones obtained by Desrayud and Fichera [3]: a qualitatively one comparing streamlines and isotherms and a quantitatively one comparing the protruding heater module temperature achieved. Both streamlines and isotherms had the same pattern compared to Desrayud and Fichera's [3] results as shown in Figs. 2 and 3.

Finally, the quantitively validation is shown in Tab. 3. The maximum percentage difference was 5.21% for the case where $A = 12.5$ and $Ra = 10^6$. The temperatures from Desrayud and Fichera's work come from the correlation created by them, which has an average error of 2.5% and maximum individual error of 5% according to their work [3]. Since the maximum percentage difference obtained in the present work was also 5% and the difference for the other cases are even lower, the model created was considered as validated.

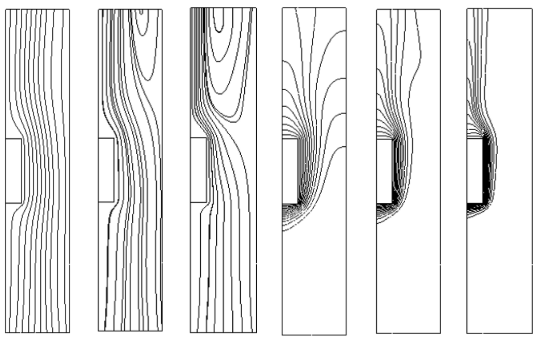

Figure 2. From left to right: streamlines $Ra = 10^4$, streamlines $Ra = 10^5$, streamlines $Ra = 10^6$, isotherms $Ra = 10^4$, isotherms $Ra = 10^5$ and isotherms $Ra = 10^6$. For all figures A = 5, h = 1 m and w = 0.00265 m. Present work.

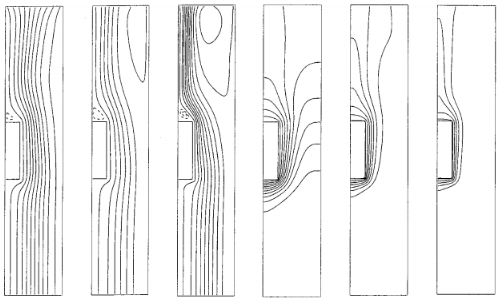

Figure 3. From left to right: streamlines $Ra = 10^4$, streamlines $Ra = 10^5$, streamlines $Ra = 10^6$, isotherms $Ra = 10^4$, isotherms $Ra = 10^5$ and isotherms $Ra = 10^6$.For all figures A = 5, h = 1 m and w = 0.00265 m. Autorship: Desrayud and Fichera [3]

.

Table 3. Heater module temperature comparison.

	Ra = 10ˆ6			Ra = 10ˆ5			Ra = 10ˆ4		
A	T	T [3]	Relative Error	T	T [3]	Relative Error	T	T [3]	Relative Error
5	307.537	308.644	0.36%	301.0745	301.1687	0.031%	300.1536	300.158	0.0015%
10	358.1197	368.6405	2.85%	308.0685	309.2807	0.392%	301.1881	301.2548	0.0221%
12.5	411.2744	433.8861	5.21%	315.5378	318.1024	0.806%	302.3	302.4476	0.0488%

3 Results

The results obtained allowed this paper to show the Rayleigh number effect on the temperature distribution along the channel and also perform a sensibility analysis regarding the emissivity of the protruding element. The Rayleigh number distribution along the channel's left side, which includes the heater module, is shown in the Fig. 4 and explicit the same tendency for all three aspect ratios considered. Regarding the protruding element maximum temperature, it rises as the Rayleigh number rises, as expected, because the heat transfer rate changes linearly as the Rayleigh number. When it comes to the Nusselt number and the convective heat transfer coefficient, both go down when the Rayleigh number becomes lower as seen in Tab 4. It might be explained due to the principles of natural convection, higher temperature gradients means higher density gradients which leads to higher velocities thanks to the buoyancy effect. The convective heat transfer coefficient is directly proportional to the flow velocity, resulting in higher convective heat transfer coefficients for higher Rayleigh numbers, and therefore, higher Nusselt numbers as well. Figure 5 shows the local heat transfer coefficient on the protruding element and an increase is noticeable on element corners due to the temperature gradient and its increase is higher in the lower corner owing to the bigger temperature gradient in that region, as one would expect.

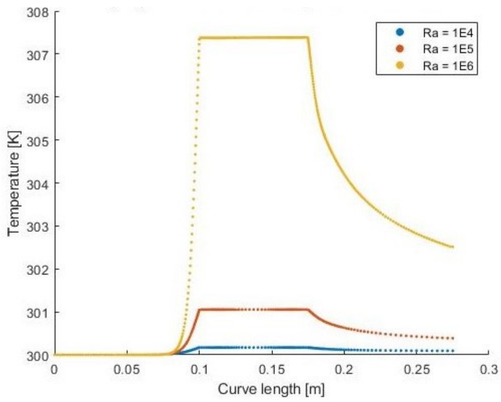

Figure 4. Temperature distribution along the channel for $A = 5$, h = 1 m and w = 0.00265 m..

The emissivity analysis shown in Fig. 6 resulted in lower maximum temperature heater module as the emissivity is higher, as one would expect. The higher the emissivity, the bigger amount of heat is dissipated through

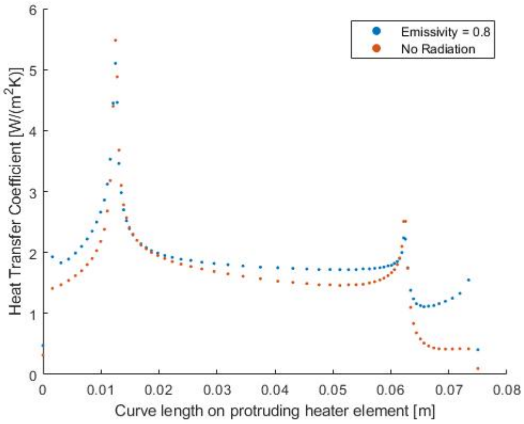

Figure 5. Heat transfer coefficient along protruding heater module for $A = 5$, Ra = 10^6, h = 1 m and w = 0.00265 m..

Table 4. Nusselt number and heat transfer coefficient variation for A = 10.

Ra	Temperature [K]	Convective heat transfer coefficient (h) [W/m^{2}K]	Nu
10^6	358.1197	8.45	12.04
10^5	308.0685	6.08	8.68
10^4	301.1881	4.13	5.89

radiation, the less is dissipated in natural convection and, therefore, the convective heat transfer coefficient and the Nusselt number are lower. This tendency was the same for all three channel aspect ratios and Rayleigh numbers used in this paper.

The maximum percentual difference comparing and emissivity of 0.04 and 0.8 was 6%, for $A = 12,5$ and $Ra = 10^6$, shown in Tab 5. The 0.8 emissivity, which represents anodized aluminum, for the $A = 5$ and $Ra = 10^4$ case makes 58% of heat being dissipated by radiation and for all cases is always higher than 30%, having an important role in the heat transfer phenomena.

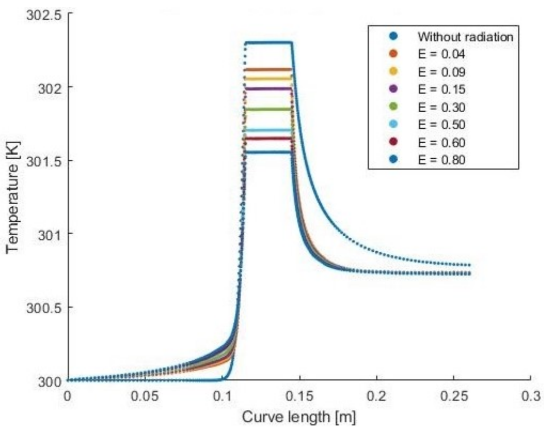

Figure 6. Emissivity effect on temperature distribution for A = 12.5, Ra = 10^5, w = 0.005 m and h = 0.02 m.

Table 5. Emissivity sensitivity analysis for A = 12.5, Ra = 10^6, w = 0.005 m and h = 0.02 m.

Emissivity	Temperature [K]	Total Heat Flux (W)	Radiation Heat Flux [W]	Convection Heat Flux [W]	h [W/m^{2}K]	Nu
0.04	397.5	35.9600	1.0332	34.9268	11.9408	13.6207
0.09	394.8396	35.960	2.2039	33.7561	11.8643	13.5334
0.15	391.9878	35.960	3.4608	32.4992	11.7766	13.4334
0.3	386.0865	35.960	6.0670	29.8930	11.5748	13.2032
0.5	380.0941	35.960	8.7182	27.2418	11.3374	12.9324
0.6	377.6532	35.960	9.7988	26.1612	11.2299	12.8098
0.8	373.5568	35.960	11.6123	24.3477	11.0335	12.5857

4 Conclusions

In this work, a numerical study of radiation heat transfer on natural cooling over a single heat source mounted in a channel was investigated numerically using a finite volume method. The numerical model without radiation was validated with literature. For the laminar flow condition, the effect of radiation heat transfer mechanism on natural convection cooling was investigated for a range of emissivity from 0.04 to 0.8. The radiaton has indeed an important role on heat transfer when it comes to natural convection driven flows and high emissivity surfaces. It was found that the heat transfer rate varied between 7.7% and 58.5% representing the lowest and highest emissivity value, respectively. Therefore, choosing materials with high emissivity when projecting a natural convection cooling system should be a concern when higher cooling efficiencies are needed and it is not desirable to use an external power source to promote convection.

Acknowledgements. The authors acknowledge the Federal University of ABC for the financial support.

Authorship statement. The authors hereby confirm that they are the sole liable persons responsible for the authorship of this work, and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors, or has the permission of the owners to be included here.

References

[1] Rocha, A. D., 2006. *Convecção Natural em Placa nas Posições Vertical e Inclinada Contendo Elementos Protuberantes Aquecidos*. PhD thesis, Faculdade de Engenharia Mecânica da Universidade Estadual de Campinas, Campinas, Brasil.
[2] Kang, B. H. & Jaluria, Y., 1990. Natural convection heat transfer characteristics of a protruding thermal source located on horizontal and vertical surfaces. *Journal Heat Mass Transfer*, vol. 3, pp. 1347–1356.
[3] Desrayud, G. & Fichera, A., 2003. On natural convective heat transfer in vertical channels with a single surface mounted heat-flux module. *Journal Heat Transfer*, vol. 125, pp. 734–739.
[4] Avelar, A. C. & Ganzarolli, M. M., 2001. *Convecção Natural em Canais Verticais Contendo Elementos Protuberantes: Aquecimento Uniforme e Não Uniforme das Placas*. PhD thesis, Faculdade de Engenharia Mecânica da Universidade Estadual de Campinas, Campinas, Brasil.
[5] Hung, Y. H.and Shiau, W. M., 1988. Local steady-state natural convection heat transfer in vertical parallel plates with two-dimensional rectangular rib. *Journal Heat Mass Transfer*, vol. 31, pp. 1279–1288.
[6] Said, S. A. M. & Krane, R. J., 19. An analytical and experimental investigation of natural convection heat transfer in vertical channels with a single obstruction. *Journal Heat Mass Transfer*, vol. 33, pp. 1121–1134.
[7] Sarper, B., Saglam, M., Aydin, O., & Avci, M., 2017. Natural convection heat transfer from discretely heated vertical channel: Effect of the surface radiation. In *Proceeding of the 2º Wolrd Congress on Momentum, Heat and Mass Transfer.*, pp. ENFHT 106–1 – ENFHT 106–8, Barcelona, Spain. International Academy of Science, Engineering and Technology.
[8] Tkachenko, S., Timchenko, V., Yeoh, G., & Reizes, J., 2019. Effects of radiation on turbulent natural convection in channel flows. *International Journal of Heat and Fluid Flow*, vol. 77, pp. 122–133.
[9] Andreozzi, A. & Manca, O., 2015. Radiation effects on natural convection in a vertical channel with an auxiliary plate. *International Journal of Thermal Sciences*, vol. 97, pp. 41–55.
[10] Incropera, F. P. e. a., 2008. *Fundamentos da Trnasferência de Calor e Massa*. Livros Técnicos e Científicos Editosa S.A.
[11] Celik, I. B., Ghia, U., Roache, P. J., Freitas, C. J., Coleman, H., & Raad, P. E., 2008. Procedure for estimation and reporting of uncertainty due to discretization in cfd applications. *Journal of Fluids Engineering*, vol. 130.