The current issue and full text archive of this journal is available at
www.emeraldinsight.com/0961-5539.htm

Received 21 September 2006
Revised 7 June 2007
Accepted 7 June 2007

Numerical analysis of turbulent natural convection heat transfer inside a triangular-shaped enclosure utilizing computational fluid dynamic code

M. Ghassemi
Faculty of Mechanical Engineering, K.N. Toosi University of Technology, Tehran, Iran
M. Fathabadi
Islamic Azad University, Alīgūdarz, Iran, and
A. Shadaram
Faculty of Mechanical Engineering, K.N. Toosi University of Technology, Tehran, Iran

Abstract

Purpose – The paper's purpose is to consider a numerical study of turbulent natural convection heat transfer inside a triangular-shaped enclosure.

Design/methodology/approach – In the formulation of governing non-linear partial differential equations the momentum and energy equations coupled with a k-ε model are applied to the enclosure. To solve these equations, a commercially available computational fluid dynamic (CFD) code, Fluent, is utilized. In addition a control volume-based code is developed. Finally, the results are compared.

Findings – Flow and temperature field are presented as a function of aspect ratio (Ar), angle between the sloped and horizontal wall (θ) and the Grashof number (Gr). It is shown that heat transfer is higher for turbulent flow when compared with laminar flow. Meanwhile the results reflect a strong dependency on the angle between two enclosure walls (θ). It is clear from the data that the results obtained by CFD code are similar to that of control volume method.

Research limitations/implications – The case considered is two-dimensional, the motion is two-dimensional and steady state, the flow is incompressible, the flow is Boussinesq, and the fluid properties are constant. It is recommended to conduct an experimental test in order to validate the analytical results.

Originality/value – The code enables the prediction of the heat transfer inside an attic-shaped enclosure. This helps in locating the highest area of heat loss; hence prevention can be implemented for this area.

Keywords Convection, Turbulent flow, Fluid dynamics

Paper type Research paper

International Journal of Numerical Methods for Heat & Fluid Flow
Vol. 18 No. 1, 2008
pp. 14-23
© Emerald Group Publishing Limited
0961-5539
DOI 10.1108/09615530810839714

Nomenclature

T	= temperature	g	= acceleration
ρ	= density	U	= velocity, x direction
ρ_o	= density of boundary layer	V	= velocity, y direction

P	= pressure	T_{H}	= temperature, hot wall
ν	= kinematic viscosity	Ra	= Rayleigh number
ν_{t}	= turbulent kinematic viscosity	K	= thermal conductivity
μ	= viscosity	k	= turbulent kinetic energy
μ_{t}	= turbulent viscosity	ε	= turbulent dissipation
Re_{t}	= turbulent Reynolds number	Div	= divergence
Pr	= Prandtl number	Grad	= gradient
Gr_x	= local Grashof number, $g\beta(\theta_{\mathrm{H}} - \theta_{\mathrm{c}}) H^3/\nu^2$	φ	= general dependent variable
		Γ_{t}	= turbulent diffusion coefficient
Nu_x	= local Nusselt number, $Nu = h_x/k_f$	β	= coefficient of thermal expansion
Gr	= average Grashof number	Ar	= aspect ratio, H/D
Nu	= average Nusselt number	A	= area
T_{C}	= temperature, cold wall	S_{φ}	= general source term

Introduction

The onset of natural convection is predicted by the Grashof number (Kakac, 1985). However, for an enclosure, the Rayleigh number is usually used. Meanwhile theoretical and experimental analysis of natural convection in an enclosure is available for a wide range of geometry and orientation of the enclosure, Grashof numbers and aspect ratio (Prodip and Shohel, 2002; Shohel *et al.*, 2002; Oosthuizen, 1999; Fedrov and Viskanta, 1997; Bejan, 1984; Flack *et al.*, 1979).

Flack has experimentally measured the velocity and heat transfer inside a triangular shaped attic using Rayleigh number as high as 10^6 (Flack *et al.*, 1979, Flack, 1980). However, Bejan experimentally determined the heat transfer inside an attic enclosure for Rayleigh numbers as high as 10^9 (Poulikakos and Bejan, 1983). Laminar natural convection inside a triangular shaped enclosure cooled and heated from below was investigated by Ghassemi and Roux (1989). Ghassemi obtained the two dimensional, steady state laminar solution for a range of aspect ratios (0.2-0.9), and the Grashof numbers (10^4-10^6). However, this study examines turbulent natural convection inside a triangular enclosure heated from below. In second study by Ghassemi *et al.* (2003) the turbulent natural convection inside a triangular enclosure for winter situation where the sloped wall temperature is much lower than horizontal wall was investigated. In this study, Ghassemi and Fathabadi presented the effect of a high-Grashof number ($Gr > 10^6$) and aspect ratio on heat transfer by using a finite difference control volume-based method. The purpose of this study is to utilize a computational fluid dynamic (CFD) based program, Fluent, which is commercially available to obtain heat transfer results as a function of angle (θ) and aspect ratio (Ar). Furthermore, we verify the results with data that is calculated by control volume-based developed program.

Problem statement

The vertical boundary is treated physically as a plane of symmetry and only one half of the triangular shaped enclosure about the plane of symmetry is investigated (Figure 1).

The horizontal wall is kept at a higher uniform temperature relative to the sloped wall so that the Grashof number is always greater than the transient Grashof number ($Gr > 10^6$).

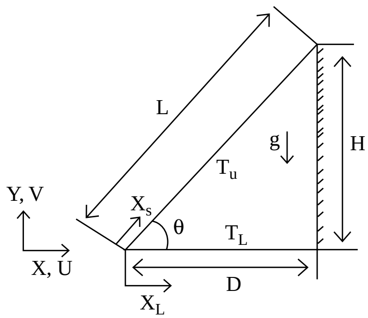

Figure 1.
Schematic illustration of a triangular enclosure

The governing equations with assumptions:

(1) the motion is two dimensional and steady state;
(2) the flow is incompressible;
(3) Boussinesq; and
(4) the fluid properties (ν, k, α) are constant

are:
Continuity equation:

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

X-momentum equation:

$$\frac{\partial(\rho uu)}{\partial x}+\frac{\partial(\rho uv)}{\partial y}=-\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left[(\mu+\mu_t)\frac{\partial u}{\partial x}\right]+\frac{\partial}{\partial y}\left[(\mu+\mu_t)\frac{\partial u}{\partial y}\right] \tag{2}$$

Y-momentum equation:

$$\frac{\partial(\rho uv)}{\partial x}+\frac{\partial(\rho vv)}{\partial y}=-\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left[(\mu+\mu_t)\frac{\partial v}{\partial x}\right]+\frac{\partial}{\partial y}\left[(\mu+\mu_t)\frac{\partial v}{\partial y}\right]+g\beta(T-T_C) \tag{3}$$

Energy equation:

$$\frac{\partial(\rho uT)}{\partial x}+\frac{\partial(\rho vT)}{\partial y}=\frac{\partial}{\partial x}\left[\left(\frac{k}{C_p}+\frac{\mu_t}{Pr_t}\right)\frac{\partial T}{\partial x}\right]+\frac{\partial}{\partial y}\left[\left(\frac{k}{C_p}+\frac{\mu_t}{Pr_t}\right)\frac{\partial T}{\partial y}\right] \tag{4}$$

Turbulent kinetic energy equation (K) (Ghassemi *et al.*, 2003):

$$\frac{\partial(\rho uK)}{\partial x}+\frac{\partial(\rho vK)}{\partial y}=\frac{\partial}{\partial x}\left[\left(\mu+\frac{\mu_t}{6_k}\right)\frac{\partial K}{\partial x}\right]+\frac{\partial}{\partial y}\left[\left(\mu+\frac{\mu_t}{6_k}\right)\frac{\partial K}{\partial y}\right]-\rho\varepsilon+G \tag{5}$$

Turbulent dissipation equation (ε) (Shohel *et al.*, 2002):

$$\frac{\partial(\rho u\varepsilon)}{\partial x}+\frac{\partial(\rho v\varepsilon)}{\partial y}=\frac{\partial}{\partial x}\left[\left(\mu+\frac{\mu_t}{6_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x}\right]+\frac{\partial}{\partial y}\left[\left(\mu+\frac{\mu_t}{6_\varepsilon}\right)\frac{\partial \varepsilon}{\partial y}\right]+C_1 f_1\left(\frac{\varepsilon}{k}\right)G$$

$$-C_2 f_2\rho\frac{\varepsilon^2}{k} \tag{6}$$

where G and turbulent viscosity (μ_t) are as follows:

$$G = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} \quad (7)$$

$$\mu_t = \rho C_\mu f_\mu \left(\frac{K^2}{\varepsilon} \right) \quad (8)$$

Other constants (i.e. C_1,C_2, etc.) are given in Ghassemi *et al.* (2003).

Boundary conditions

The following boundary conditions must be satisfied by the above equations:

$$y = 0, 0 < x < D, u = v = 0, T = T_C, K = 0, \varepsilon = 2\left(\frac{\mu}{\rho}\right)\left(\frac{\partial k^{1/2}}{\partial x}\right)^2 \quad (9a)$$

$$x = D, 0 < y < H, Q = 0, \quad \frac{\partial T}{\partial x} = 0, \quad \frac{\partial v}{\partial x} = 0, \quad \frac{\partial \varepsilon}{\partial x} = 0 \quad (9b)$$

$$Y = F(x), x = G(y), u = v = 0, T = T_C, K = 0, \varepsilon = 2\left(\frac{\mu}{\rho}\right)\left(\frac{\partial k^{1/2}}{\partial x}\right)^2 \quad (9c)$$

where:

$$G(x) = \frac{xH}{D} \quad \text{and} \quad F(y) = \frac{yD}{H}$$

Solution outline

CFD-based Fluent, is used to solve equations (2)-(6). Furthermore, equations (2)-(6) are written in a general form as given by equation (11) and then Patankar's SIMPLER finite difference, control volume scheme is applied (Ghassemi *et al.*, 2003):

$$\mathrm{div}(\rho\phi U) = \mathrm{div}[\Gamma \mathrm{grad}\phi] + S_\phi \quad (10)$$

The ϕ Γ_ϕ and S_ϕ are determined and then the equations are solved simultaneously. The convection terms are discretized by using hybrid differencing, which includes central and upward differencing. The resulting discretized equations are solved by a line-by-line solution method in conjunction with the tri-diagonal matrix algorithm. The maximum residual of mass, momentum and energy over all nodes of computation domain are selected to be 0.001. With this limitation, it is observed that the maximum number of iteration for each run of the program is around 300. Results obtained by Fluent program are finally compared with the results obtained by control volume-based method.

Heat transfer equations

The most important parameters are the heat transfer and Nusselt number. The local Nusselt number along the horizontal wall and the heat flux along the sloped wall is given by, respectively (Ghassemi *et al.*, 2003):

$$Nu_y = \frac{\left.\frac{\partial T}{\partial y}\right|_{y=0} H}{T_{\mathrm{H}} - T_{\mathrm{C}}} \tag{11}$$

and the heat flux along the sloped wall is given by (Ghassemi and Roux, 1989):

$$q = \sqrt{\left(-k\frac{\partial T}{\partial x}\right)^2 + \left(-k\frac{\partial T}{\partial y}\right)^2} = k\frac{\partial T}{\partial N} \tag{12}$$

or:

$$q = h(T_{\mathrm{H}} - T_{\mathrm{C}}) \tag{13}$$

where:

$$\frac{\partial T}{\partial N} = \sqrt{\left(\frac{\partial T}{\partial x}\right)^2 + \left(\frac{\partial T}{\partial y}\right)^2}$$

Numerical result and discussion

The accuracy of the control volume-based numerical results is verified by a CFD-based code called "Fluent," as shown in Figure 2. The CFD-based results follow the same trend as data obtained by control volume-based program (Ghassemi *et al.*, 2003).

Figure 3 shows the non-dimensional centerline temperature profile along the vertical plane of symmetry. As shown, there is no isotherm in the turbulent case as opposed to the laminar case. The temperature gradient is uniform and resembles a straight line. Also, as the aspect ratio increases, the slope of temperature gradient line decreases.

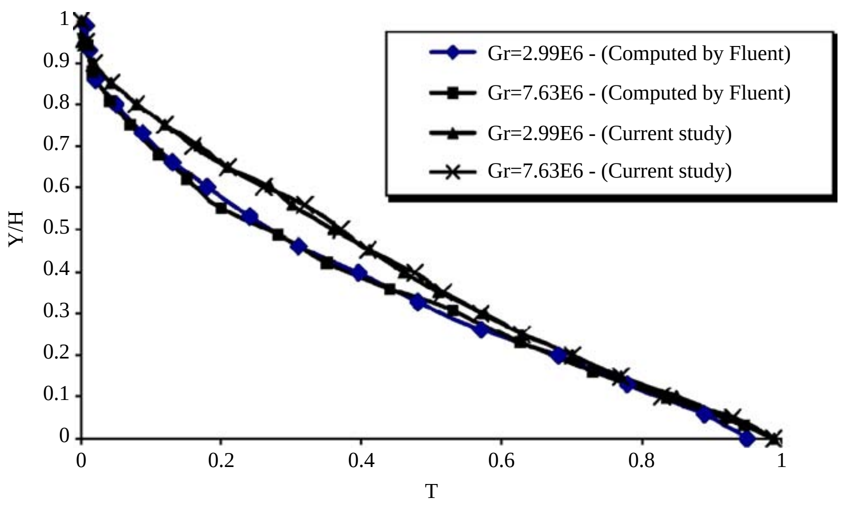

Figure 2.
Non-dimensional centerline temperature, Ar = 1

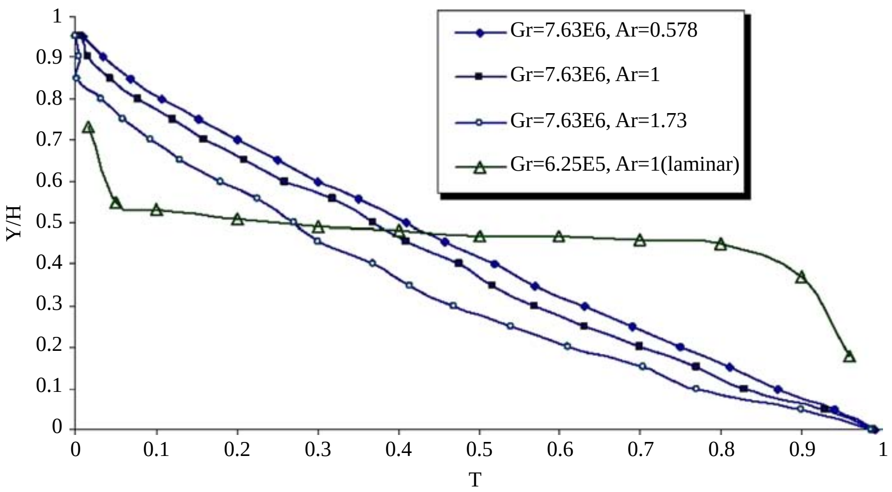

Figure 3.
Non-dimensional centerline temperature

Figure 4 shows the flow pattern inside the triangular enclosure. As shown, the flow moves downward along the sloped wall due to gravity that pulls the cooler (higher density) fluid downward. On the other hand the velocity in the bottom corner, where the cold and hot walls meet, decreases and approaches zero due to higher viscous forces. The velocity is highest near the vertical plane of symmetry where there is a recirculation area near the bottom (hot) wall as shown in Figure 3.

The heat flux along the sloped wall is shown in Figures 5 and 6. Figure 5 shows the heat distribution for aspect ratio of unity while Grashof number changes. As shown in the figure increasing the Grashof number increases the heat flux along the sloped wall

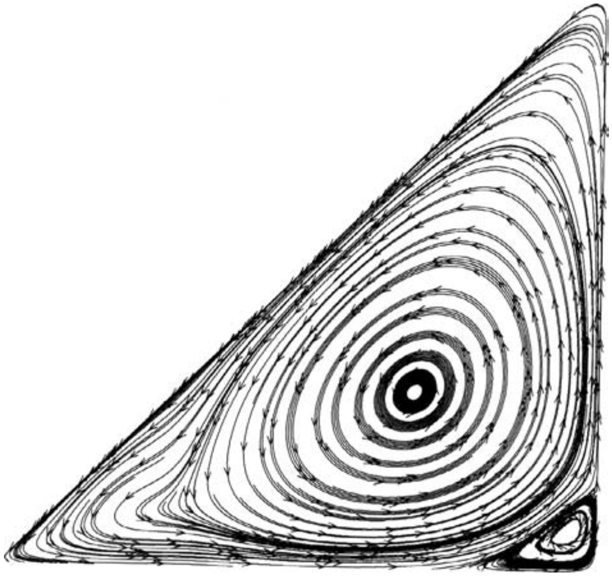

Figure 4.
Velocity field illustration ($\theta = 45°$ and $Gr_t = 8.8 \times 10^5$)

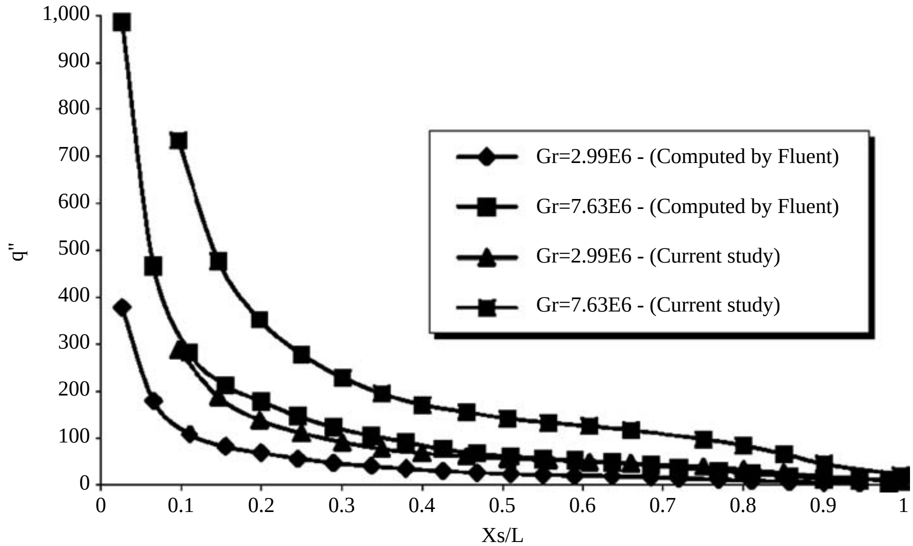

Figure 5.
Heat flux distribution along the sloped wall, Ar = 1

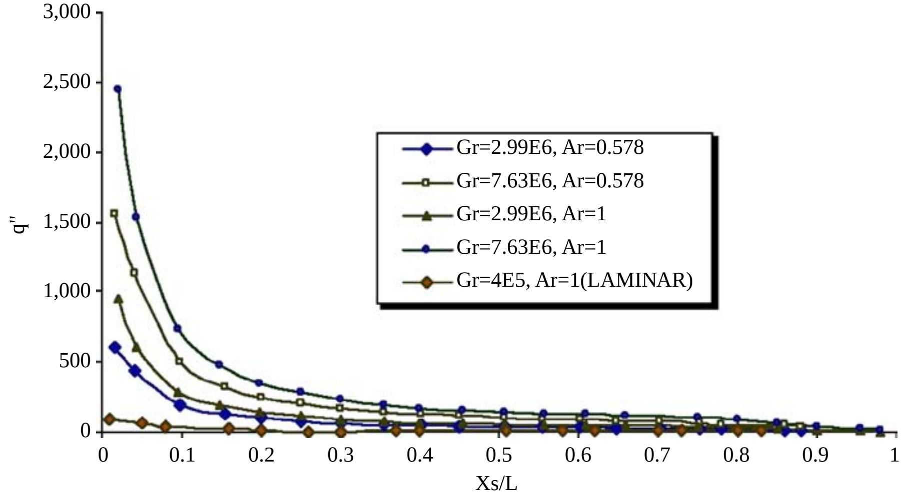

Figure 6.
Heat flux distribution along the sloped wall

for both methods. Likewise, the CFD-based results follows the same trends as control volume-based data except for area near the corner.

Figure 6 shows that heat flux along the sloped wall is directly affected by the aspect ratio. As shown by the Figure 6, for a given Grashof number, increasing the aspect ratio increases the heat flux along the sloped wall. In general, heat flux on sloped wall decreases as it moves away from the corner where the two walls meet. At the top of the enclosure, there is a sharp drop in the heat flux and the amount of this drop is almost the same as in the laminar case.

Figures 7 and 8 show the heat flux along the horizontal wall. Again for a fixed aspect ratio (Ar = 1) heat flux increases as the Grashof number increases, see Figure 7.

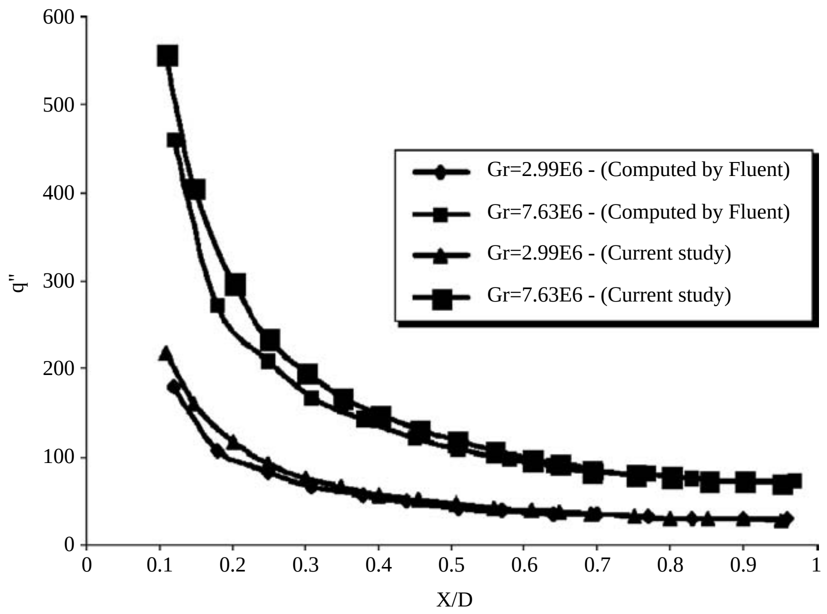

Figure 7.
Heat flux distribution along the horizontal wall, Ar = 1

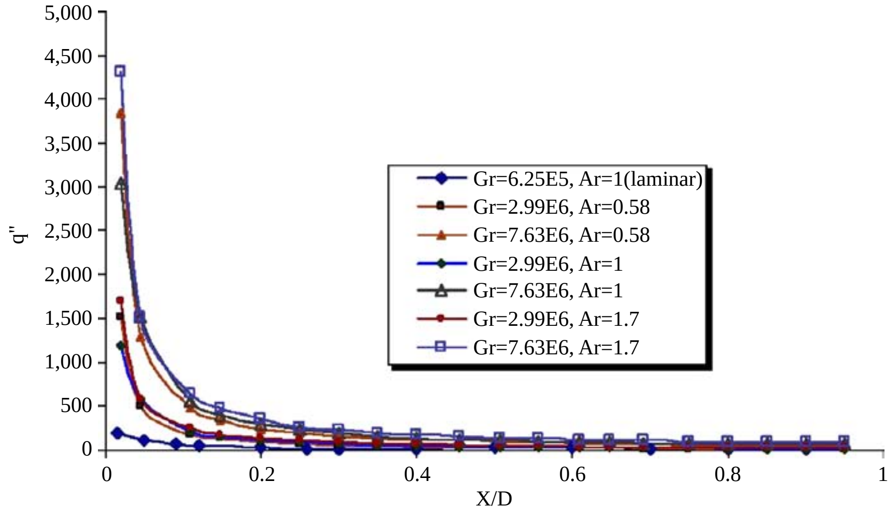

Figure 8.
Heat flux distribution along the horizontal wall

Figure 8 shows the behavior of heat flux along the horizontal wall. As shown, increasing the aspect ratio increases the heat flux along the horizontal wall. Again, heat flux on horizontal wall decreases as it moves away from the corner where the two walls meet. At the top of the enclosure, there is a sharp drop in the heat flux and the amount of this drop is almost the same as that found in the laminar case.

Figure 9 shows the effect of the Grashof number and angle θ (30 and 45) on heat flux at the midpoint of the sloped wall. As shown, the Grashof number increases as the heat flux at the midpoint increases. Also, at a fixed Grashof number, the heat flux increases as the angle increases.

The dependence of the Nusselt number on the local Grashof number for different aspect ratios is shown by Figure 10. As shown in the figure, the Nusselt number decreases as the local Grashof number increases (Figure 10). However, at a given Grashof number, as the aspect ratio increases the Nusselt number increases slightly.

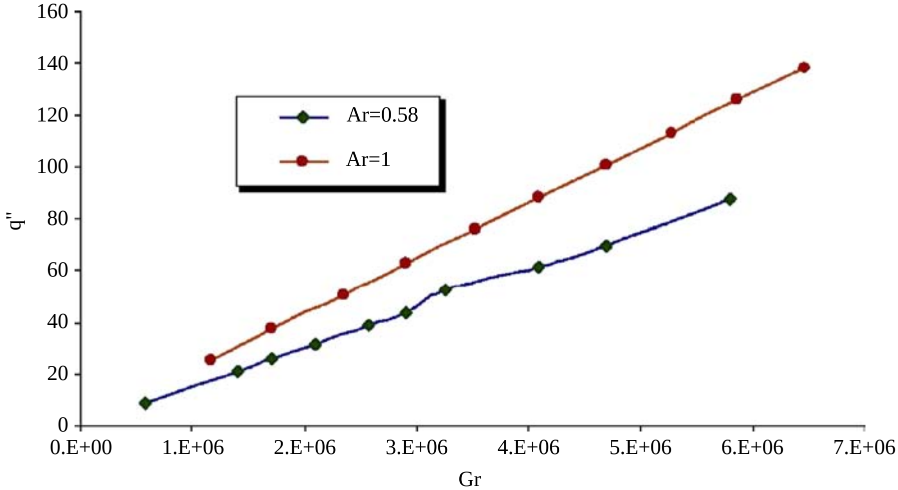

Figure 9.
Heat flux at horizontal wall midpoint as a function of the Grashof number

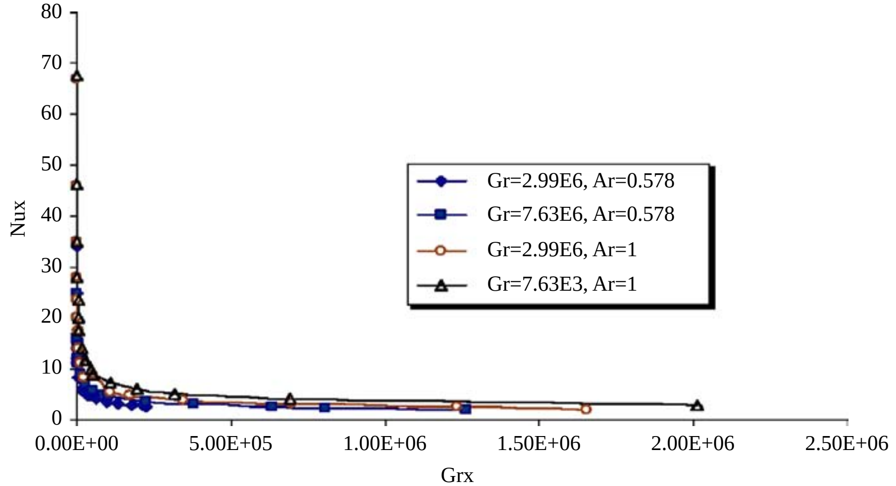

Figure 10.
Local Nusselt number along the sloped wall as a function of the Grashof number

References

Bejan, A. (1984), *Convective Heat Transfer*, Wiley, New York, NY.

Fedrov, A.G. and Viskanta, R.V. (1997), "Turbulent natural convection heat transfer in an asymmetrically heated vertical parallel – plate channel", *International Journal of Heat & Mass Transfer*, Vol. 40 No. 16, pp. 3849-60.

Flack, R.D. (1980), "The experimental measurement of natural convection heat transfer in enclosure heated or cooled from below", *ASME Journal of Heat Transfer*, Vol. 102, pp. 770-2.

Flack, R.D., Konopnicki, T.T. and Rooke, J.H. (1979), "The measurement of natural convective heat transfer in triangular enclosure", *ASME Journal of Heat Transfer*, Vol. 101 No. 4, pp. 648-54.

Ghassemi, M. and Roux, J.A. (1989), "Numerical investigation of natural convection within a triangular shaped enclosure", *National Heat Transfer Conference, Heat Transfer in Convective Flows*, Vol. 107, pp. 169-75.

Ghassemi, M., Keshavarz, A. and Fathabadi, J. (2003), "Numerical investigation of turbulent natural convection in a triangular shaped enclosure", paper presented at the 6th ASME-JSME Thermal Engineering Joint Conference, pp. 1-7.

Kakac, S. (1985), *Natural Convection*, Hemisphere Publishing Corporation, New York, NY.

Oosthuizen, P.H. (1999), "Unsteady free convection flow in enclosure with a stepwise periodically varying side-wall heat flux", *International Journal of Numerical Methods for Heat & Fluid Flow*, Vol. 9 No. 3, pp. 214-23.

Poulikakos, D. and Bejan, A. (1983), "Numerical study of transient high Rayleigh number convection in an attic-shaped porous layer", *Trans. ASME Journal of Heat Transfer*, Vol. 105, pp. 476-84.

Prodip, K.D. and Shohel, M. (2002), "Numerical investigation of natural convection inside a wavy enclosure", *International Journal of Thermal Sciences*, Vol. 22 No. 1, pp. 34-9.

Shohel, M., Prodip, K.D., Nasim, H. and Sadrul Islam, A.K.M. (2002), "Free convection in an enclosure with vertical wavy walls", *International Journal of Thermal Sciences*, Vol. 5 No. 41, pp. 440-6.

Further reading

Flack, R.D. and Witt, C. (1979), "Velocity measurements in two-dimensional natural convection air flows using a laser velocimeter", *ASME Journal of Heat Transfer*, Vol. 101 No. 2, pp. 256-60.

Oosthuizen, P.H. (1999), "A numerical study of three-dimensional natural convection in a low aspect ratio horizontal enclosure with a linearly varying side-wall temperature", *Proc. 7th Annual Conf. of the CFD Society of Canada*, pp. 4/45-4/50.

Corresponding author

M. Ghassemi can be contacted at: ghasemi@kntu.ac.ir

To purchase reprints of this article please e-mail: **reprints@emeraldinsight.com**
Or visit our web site for further details: **www.emeraldinsight.com/reprints**