International Journal of Thermal Sciences 125 (2018) 176–184

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

journal homepage: www.elsevier.com/locate/ijts

Forced convective heat transfer and flow characteristics of fractal grid heat sinks

Shiva Skanthan[a], Chin Vern Yeoh[a], Wai Meng Chin[b], Ji Jinn Foo[a,*]

[a] School of Engineering, Monash University Malaysia, 47500 Bandar Sunway, Selangor, Malaysia
[b] Daikin R&D Sdn Bhd, 47000 Sungai Buloh, Selangor, Malaysia

ARTICLE INFO

Keywords:
Forced convection
Parallel flow
Fractal geometry
Heat sink
Thermal dissipation

ABSTRACT

The continual miniaturization of electronic components has led to increasing levels of thermal dissipation rate per unit volume. The present 3D numerical study proposes a novel method to enhance thermal dissipation via forced convective heat transfer with a 2D planar space-filling insert. Effects of grid curvature were investigated by comparing a circular fractal grid (CFG) to a square fractal grid (SFG) and a control regular grid (RG) of approximately equivalent blockage ratio. The fractal configurations employed possess similar thickness ratios to ensure an integrated comparison based on the interstitial curvatures of CFG and SFG. Heat flux of 20×10^3 W/m^2 was applied along the four sides of each insert at Reynolds number of $Re_h = 2.04 \times 10^4$. The flow field and thermal dissipation were solved numerically with the Reynolds Stress Model (RSM) and standard energy equation. Results show that the sharp curvature discontinuities and higher exposed surface area in the SFG leads to a higher Nusselt number. While the SFG has, a higher pressure drop, ΔP, as compared to CFG and RG, the enhancement in forced convective heat transfer offsets the higher ΔP, which results in a higher overall system performance. By this measure of performance, the SFG and CFG outperforms the RG by 35% and 9%, respectively. In terms of turbulent mixing, the CFG achieves higher turbulent intensities leeward from the grid than that of SFG and RG. This suggests the pivotal role of curvature in enhancing the mixing properties of a space-filling insert.

1. Introduction

One of the unique features of turbulence is diffusivity, which plays a significant role in the enhanced mixing and increased rate of mass, momentum and energy transport in a flow. Turbulent mixing plays a significant role in many industrial engineering applications such as electronic cooling, chemical mixing and automotive aerodynamics. At higher Reynolds numbers, a considerable number of eddies are generated, spanning across various characteristic length scales viz., integral length scale, Taylor microscale and the Kolmogorov length scale. Originally, quasi-isotropic turbulence can be produced via passive means in a wind and water tunnel using planar regular grids [1]. However, less than a decade ago, a new idea was proposed by introducing planar fractal geometry to generate turbulence due to its ability in producing higher turbulence intensities than those achieved using a regular grid [2]. Various distinct and desired turbulent fluctuations can be attained using fractal planar geometry due to its ability to rescale upstream flow into an array of frequencies [3].

Plate and pin fins are widely utilized as heat sinks in the cooling of HVAC systems, graphical processing units (GPU's) and microprocessors. This is in part due to the increase in available surface area for thermal dissipation. Other methods such as synthetic jet air cooling [4,5] and the use of phase change materials/thermosyphons which can store a great deal of energy due to their heat of fusion [6] have also been attempted. However, the continual miniaturization of electronic components has led to increasing levels of thermal dissipation rate per unit volume. Hence, effective and efficient heat transfer remains a critical goal in thermo-fluid engineering in a bid to provide more reliable electronic devices and HVAC systems for a rapidly demanding consumer market. This implies a need for higher heat transfer rates with lower levels of pressure drop. This brings about a few compelling questions, viz.: Is an effective heat transfer process primarily dependent on heat dissipation surface area and fluid flow rate? Are there any other methods to promote thermal exchange, such as improving the quality of fluid flow through turbulence generation, leading to enhanced mixing and heat transfer rates? The latter motivation was studied by Gori and Petracci in a bid to determine the effect of the turbulence intensity level on the convective heat transfer of a slot on a jet-impinging cylinder.

---

[*] Corresponding author.
E-mail address: Foo.Ji.Jinn@monash.edu (J.J. Foo).

https://doi.org/10.1016/j.ijthermalsci.2017.11.016
Received 19 May 2017; Received in revised form 18 August 2017; Accepted 20 November 2017
1290-0729/

They found a sensitive increment in turbulence intensity level due to the presence of the grid located between the slot and the cylinder with localized Nusselt numbers increasing up to 10% at $S/H < 4$, with S being the distance of the cylinder from the slot exit and H the slot height. However, the effect of turbulence generated by the grid is completely dissipated beyond a streamwise location of 10 slot widths, reducing the working area of efficacy of such a system [7]. Since then there has been a push to determine better methods to generate turbulence passively. The introduction of a 2D fractal grid as a method for passive turbulent control began to take shape through the pioneering study done by Hurst and Vassilicos [8]. A fractal grid is an arrangement of a particular geometry that rehashes and lessens in size, in this manner forming patterns of various iterations [9]. Several properties are needed to design a fractal grid, such as the number of fractal iterations, N, the thickness ratio, t_r, and the blockage ratio, σ. The thickness ratio, t_r, is the ratio between the thickness of the first iteration and the last iteration, while the blockage ratio, σ, is defined as the ratio of the grid total area to the cross-sectional area of the channel [8]. An increase in thickness ratio, t_r results in a higher turbulent intensity, however, it will also result in a larger pressure coefficient. Consequently, a proper balance of the two outcomes must be considered when designing the fractal. A key derived parameter from these base geometrical properties is the blockage ratio, with simple intuition dictating that a higher blockage ratio results in a higher pressure drop.

While the geometries of fractal grids are complicated due to such grids having structures with multiple length scales, further study by Mazellier and Vassilicos showed that the location of the peak turbulent intensity is primarily determined by the scale of the largest grid bar or first iteration of the fractal [10]. Later experimental studies showed that fractal geometries produce a long region of downstream turbulence evolution, which contradicts the Richardson-Kolmogorov cascading turbulence model [11]. Seoud and Vassilicos through experimental studies showed that fractal square grids generate an unusually high Taylor microscale, λ, scaled Reynolds number Re_λ, due to the increased turbulent intensities [12]. These intensities are about 3 × larger than those generated by classical grids [8]. Mazellier and Vassilicos [10] attempted to scale the streamwise profile of the relative intensity of the fluctuating velocity by introducing the wake interaction length X^* defined as,

$$X^* = \frac{L_0^2}{t_0} \tag{1}$$

whereby L_o and t_o are the mesh length and the width of the largest bar, respectively. The investigations done by direct numerical simulations (DNS) have been able to confirm many of the important properties discovered in earlier experimental studies [3,13–18].

Recently, there has been a push to implement fractal geometries into various practical applications. Initially, studies by Chen et al. [19] and Zhang et al. [20] have shown the promise of fractal tree-shaped networks in both single-phase and multiphase heat transfer, with increases of performance coefficients of more than unity, better temperature uniformity across the studied heat sinks, and with relatively lower pumping power. On a related note, external flow past grids with such geometries have also been shown to have good heat transfer performance. The use of fractal grids within the nozzle to tune the upstream turbulence of the incoming jet to enhance heat transfer in impinged jets was first suggested by Cafiero et al. [21]. Meanwhile, Teh et al. suggested the use of fractal grid within a T-duct to strengthen the thermal mixing of a HVAC free cooling/heating system. Fractal geometry enhances turbulence generation due to its various iterations of decreasing size. However, much of the study of fractal-induced turbulence and its application has been based on the use of fractal grids with square geometry with variation of thickness ratio, number of iterations and other quantifiable properties, with little study done on improving the inherent baseline geometry of the fractal. One such variable is that of geometrical curvature, as traditional fractal geometries introduce regions of large curvature discontinuities due to the discrete, iterative nature of their construction. However, Cafiero et al. recently studied the effect of grid geometry to regulate the convective heat transfer of impinging jets on a heated surface. This was the first time a circular fractal geometry was introduced in a thermofluids related environment. They concluded that a compromise between highly localized convective heat transfer rate and uniformity can be achieved through the application of a circular fractal insert [22].

The present investigation aims to determine the effectiveness of fractal geometrical curvature on turbulent mixing by introducing a circular fractal grid (CFG) along with a square fractal grid (SFG) and a control regular grid (RG). Furthermore, this study attempts to work on a novel heat transfer method, i.e., to employ grid inserts as a mean of thermal dissipation via forced convective heat transfer. Numerical simulation is conducted using the Reynolds Stress Model (RSM) to enhance the understanding of flow fluctuations and its effects on thermal mixing of the three grid inserts, namely: CFG, SFG and RG. The simulation is done using RSM as opposed to other methods such as Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS) largely due to its efficiency in solving the problem at hand. In addition, this study has been set up as a suggested protocol for application of grid generated turbulence study in industrial research, where simulation time and cost play an imperative role in research and development. The following study is divided as such: Section 2 describes the simulation setup and discusses in detail the geometry of the space-filling grids inserts employed, as well as the conditions for the numerical simulation. In Section 3, results are presented in terms of turbulent intensity, Nusselt number and pressure drop. An integrated performance index, η, is introduced to analyse the overall system performance of the inserts being used with respect to industrial application. Lastly, in Section 4 conclusions are drawn from the numerically simulated results and recommendations of use of fractals curvature in various applications are discussed.

2. Methodology

The present study investigates the steady-state and parallel flow characteristics, as well as the thermal dissipating performance of 2D planar space-filling fractal grids and a regular grid insert to be used as a control. The inserts are placed in a square duct to determine the effects of grid curvature on forced convective heat transfer. Firstly, the three inserts comprising of (a) $N = 3$ CFG, (b) $N = 3$ SFG, and a control RG were modelled using Dassault Systèms Solidworks (version 2016, US) as shown in Fig. 1(a–c). A complete description of the three inserts can be found in Table 1. The colored area of the grids in Fig. 1(a–c) represents the 2D planar space-filling blocked areas whilst the rest are void areas to allow airflow through the inserts. These inserts were designed to fill the interior section of the computational domain and hence were 160 × 160 mm^2 with a thickness of 5 mm. It is worth mentioning that the author has designed the grids to ensure the blockage ratio for all three grids are approximately equal given geometrical constraints, furthermore the thickness ratios for the CFG and SFG are the same to isolate the effects of grid curvature alone on the flow field generated by the fractal inserts.

The completed 3D models were then imported into ANSYS-Fluent (version 16.0, US) for pre-processing. A computational domain for the test section was modelled in the form of a cuboid with a cross-section of 160 × 160 mm^2 and total length of 4.005×10^3 mm. A schematic of the computational domain is as shown in Fig. 1(d). The x-axis aligns along the width of the channel, the y-axis in parallel to the height of the channel, and the z-axis aligns streamwise of the channel. The first 2×10^3 mm is the entrance length to develop the flow followed by a 5 mm insert and a post-grid length of 2×10^3 mm.

The 3D, steady-state, incompressible turbulence of the inserts was numerically simulated using first-order discretization of the Reynolds