International Communications in Heat and Mass Transfer 84 (2017) 86–93

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

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

Heat transfer enhancement by flexible printed circuit board's deformation

C.H. Lim[a], M.Z. Abdullah[b,*], I.A. Azid[c], C.Y. Khor[d]

[a] School of Mechanical Engineering, Universiti Sains Malaysia, Engineering Campus, 14300 Nibong Tebal, Penang, Malaysia
[b] School of Aerospace Engineering, Universiti Sains Malaysia, Engineering Campus, 14300 Nibong Tebal, Penang, Malaysia
[c] Mechanical Section, Universiti Kuala Lumpur Malaysian Spanish Institute, Kulim Hi-TechPark, 09000 Kulim, Kedah, Malaysia
[d] Faculty of Engineering Technology (FETech), Universiti Malaysia Perlis (UniMAP), Level 1, Block S2, UniCITI Alam Campus, Sungai Chuchuh, 02100 Padang Besar, Perlis, Malaysia

A R T I C L E I N F O

Keywords:
Flexible printed circuit board (FPCB)
Thermal-mechanical-fluid-structure interaction
BGA
CFD
Electronic cooling

A B S T R A C T

Flexible printed circuit board (FPCB) has started replacing rigid PCB in some electronic application. However, the deflection and heat transfer of FPCB caused by flow and thermal are far more crucial compared to rigid PCB. In the current study, effects of flow and thermal induced deflection were coupled in real time. The numerical modelling was executed using FLUENT and ABAQUS, coupled online by Mesh-based Parallel Code Coupling Interface (MpCCI). The numerical findings have been validated with experimental results. Rigid PCB with same thermal properties of FPCB has been compared. The findings showed that the deflection of the FPCB does help to increase the heat transfer of the PCB. Furthermore, models with different power of heat sources along with various Reynolds numbers (*Re*) had been studied. The results exhibited that the magnitude and the trend lines' pattern of deflection divided by characteristic length changes according to thermal power. It indicates both flow and heat have significant effect to the deflection and Nusselt number (*Nu*) of FPCB. Therefore, flow and thermal should be coupled at the same time for more realistic FPCB simulation under flow environment in operating condition.

1. Introduction

The high demand of electronic devices has caused rapid growth of electronic and microelectronic industries since the past few decades, in which flexible printed circuit board (FPCB) has started replacing rigid PCB (RPCB) in some applications due to its flexibility and light weight [1,2]. Researchers have used FPCB in several applications such as fabricated temperature sensor array and flow sensor on the FPCB for thermal and flow study, mounting flexible and light weight LED on FPCB for safety used during earthquakes, applied FPCB in computer motherboard, wire wound coil, winding motor and electrodynamics bearings [1,3–8]. However, the flexible feature of FPCB also affects deformation and increases cooling effect under flow environment with heat influences.

Air is widely used for electronic cooling due to its simplicity in producing the flow for attaining the thermal design goal. However, some other aspects such as the RPCB orientation, area of heat sink, air flow rate, components size and etc. have to be designed with caution for obtaining high cooling rate. Therefore, extensive investigation has been conducted by researchers for electronic cooling. Back in 1998, Leung and Kang [9] investigated the air-cooled RPCB assembly orientation and studied the effects of copper rib's (imitate as components) dimensions and its ratio with the wind tunnel height. Their findings suggested that flat rib with large top surface area enhance heat transfer. Tso, Xu and Tou [10] conducted an experimental study on cooling of flush-mounted discrete heat sources by using water as medium. His findings indicated that the heat transfer coefficient was extensively influenced by the *Re* and weakly by the channel height.

In the past two decades, the electronic cooling study of PCB has begun to shift from pure experimental investigation to numerical modelling due to the availability of computer and computational power. Besides, numerical simulation could help to save the material/experiment setup cost and time. 2D forced convection cooling simulation on RPCB assembly was modelled to study the flow separation on 2 heated obstacles with various sizes [11]. Eveloy, Lohan and Rodgers [12] carried out the study on the natural and forced convection for electronic cooling by using computational fluid dynamic (CFD) and validated the results with experimental findings. They proposed a paint film evaporation technique for illustrating the flow impact on the RPCB. Grimes et al. [13–16] performed detailed experiments and constructed simulation models to investigate the air flow pattern from different fan mounting configuration (mounted on inlet and outlet). They concluded

* Corresponding author.
E-mail address: mezul@usm.my (M.Z. Abdullah).

http://dx.doi.org/10.1016/j.icheatmasstransfer.2017.04.004

that the flow was laminar, uniform, steady and well predicted in simulation with the fan drawing out from the system (fan at outlet).

In recent years, Lau et al. [17,18] developed thermal coupling method. In thermal-coupling method, only thermal related properties (i.e. heat transfer coefficient, film temperature from fluid domain and surface temperature from structural domain) are coupled between two domain solvers, namely FLUENT and ABAQUS. The coupling process is handled by Multi-physics Code Coupling Interface (MpCCI) software. Nevertheless, Lau only consider the thermal aspects because flow would not cause major deflection on RPCB. Leong et al. [19–22] focused on the flow induced FPCB's deflection by coupling the flow induced force and FPCB's deformation under flow environment. However, his studies were limited to air flow effects without considering thermal effects and the electronic components were replaced by Perspex blocks for simplification. In other words, those studies are only valid when FPCB assembly is in idle condition.

From the literatures, the analysis on fluid-structure interaction (FSI) has already been carried out through coupling the interface between fluid and structural domains. However, typically flow effect and heat effect are studied separately. The coupling of flow and thermal effects has not been developed for electronic cooling. Therefore, in present study, thermal and flow effects on FPCB have been coupled concurrently in the simulation. Besides, complete BGA package with 100 solder joints has been modelled in present study so that the numerical simulation is more realistic by including thermal aspects with the validation of experimental findings. In addition, RPCB with the same model settings, factors and thermal properties (only different was FPCB's and RPCB's material properties) were simulated for studying the enhancement of heat transfer caused by the deflection of FPCB. Furthermore, single BGA package has been modelled together with FPCB under various flow velocities and thermal power for investigating the influence of those factors on the FPCB's convection heat transfer and deflection.

2. Numerical modelling

2.1. Modelling tools

In this FSI problem, two main computation platforms which are FLUENT 14.0 and ABAQUS 6.12 were used to perform the analysis for fluid and structure domains, respectively. FLUENT was used as fluid solver while ABAQUS was used to simulate the FPCB's responses in structural domain. The two domains were coupled online by Mesh-based Parallel Code Coupling Interface (MpCCI) for FSI environment. In the analysis, FLUENT exported flow induce force, film temperature and heat transfer coefficient on FPCB to ABAQUS through MpCCI while ABAQUS sent back structural deformation and temperature of BGA assembly to FLUENT as illustrated in Fig. 1.

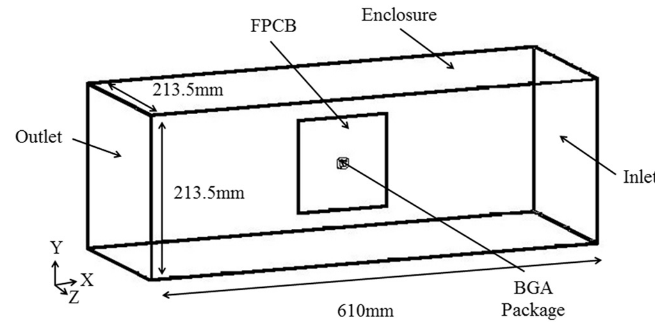

Fig. 2. Fluid domain.

2.2. Fluid flow modelling

Fluid domain with the dimensions of 213.5 mm × 213.5 mm × 610 mm which imitate wind tunnel test section had been generated in ANSYS workbench with the name selection of various boundaries as illustrated in Fig. 2. The model was meshed with 41,972 hexahedral elements with the skewness of 0.73. The number of elements is the optimum value which will be presented in the Section 2.5. The meshed model was then exported to FLUENT.

In FLUENT, three dimensional, incompressible and laminar type of flow was used. Laminar flow is ample to model wind tunnel flow because of the flow is steady and uniform as studied by Grimes [13–16]. Besides, the highest Reynolds number (*Re*) in the present study is 38,400 which is lower than the transition flow value of 2×10^5 [23]. The energy equation model in FLUENT had been activated for cases with heat source. After that, the boundary conditions were assigned as follows:

1) At enclosure and BGA assembly walls: $\vec{v} = 0$ (stationary wall with no slip condition)
2) At the inlet: $\vec{v} = \vec{v}_{desired}$ (desired flow velocity)
3) At the outlet: Pressure = 0 (ambient condition)
4) Inlet, outlet and initial temperate = 298.15 K

The present analysis involves a two-way feedback from the fluid and structure. The deformed BGA assembly structures caused by flow induced forces influence the air flow pattern and consequently influence the heat transfer. Therefore, dynamic mesh was enabled for handling the problem. In order to obtain better accuracy, second-order upwind discretization was used for the momentum and energy equation. At the end, the model was saved as case (.cas) file for following execution in MpCCI.

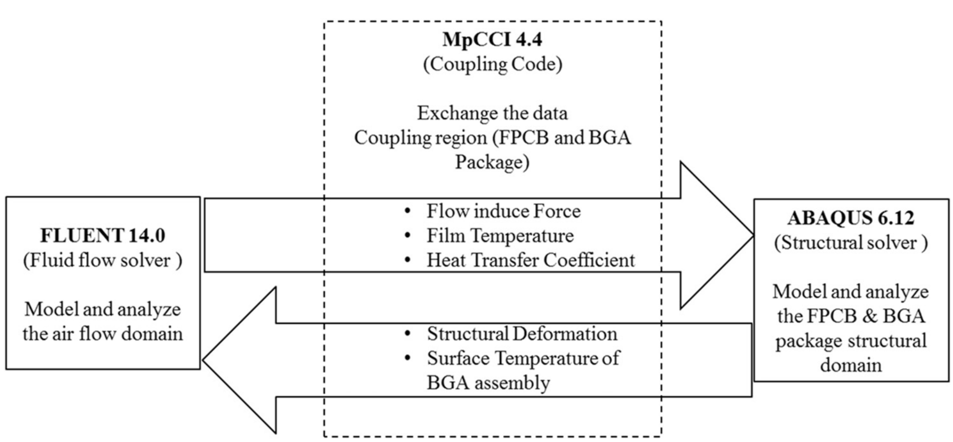

Fig. 1. MpCCI coupling.