JETIR.ORG ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

Case studies of electronics cooling

Hariom Thakur, Sinsha Prakash P K

Abstract:
As technology improves daily, investigations and studies are being done to improve the efficiency of cooling electronic devices like PCBs and the internal electronic components of projectors. The goal of this work is to give electrical gadgets a better cooling environment. To improve the heat transfer through forced convection, the location of a fan or blower installed in them is changed. When an electronic device's cooling system is constructed properly, it can quickly dissipate the high heat produced at maximum load, improving reliability. The flow behaviour is visualised using a turbulence model. This investigation is carried out using the SimFlow software with paraview. This research will help in forecasting the ideal circumstances for cooling model production for electronic systems. The three geometry files that were chosen are pins, CPU, and board.stl.Geometrics were generated in stl with X=0.0899m, Y=0.0584m, and Z=0.018m, which indicates that the blockage ratio almost ranges from 0.01to 0.05. Geometry Fan origin was created with dimensions in (m) [0.016, 0.016, 4e-03]. Since STL doesn't preserve unit information, the imported geometry is in that format.[1, 2]

Keywords: *Optimization, OpenFoam, SimFlow Electronics cooling*

Introduction:

Nowadays, electronic equipment and technologies are a part of practically every aspect of our daily life. One of these is an electronic computer, which can be anything from tiny, portable Smartphones and tablets to enormous mainframes or servers. A computer is frequently interwoven with other devices that operate it, making it difficult to tell them apart. For instance, in automobiles, satellites, spacecraft, missiles, etc. Computers are employed for a variety of tasks, from straightforward entertainment games to very complex systems supporting crucial business, academic, and military functions [3,4,5].

The most prominent examples of this are the portable electronic devices we use on a daily basis, such as laptops, cell phones, digital cameras, and other products. In these devices, a rising number of functioning components are crammed into a steadily getting smaller system box. The size of the instruments is likewise expanding. Despite getting smaller daily, the number of functions is growing. As a result, the number of functions per unit volume is dramatically rising. For desktop and server computers, compact packaging is also being created due to the need to reduce the size of the boxes and the wiring distances between electronic components.

Modern life is increasingly dominated by electronic inventions, which are becoming more compact and functional. During the preceding experiment, the heat density generated by the electric current steadily increased. An electronic chip's surface could generate heat at a rate of 100 to 10000000 watts per square metre. Large-scale electronic systems are starting to have problems with high heat flux as a result of the rise in power density. In order to keep up with the rate of development of new electronic equipment and massive electronic systems, as well as to ensure the reliability of the electronic systems, much research has been conducted to look into more effective cooling solutions.

The process of controlling the heat produced by electrical devices to stop overheating and malfunction is known as electronics cooling. When operating, electronic equipment produces heat. If this heat is not effectively dispersed, the gadget may malfunction, have a shorter lifespan, or even pose a safety risk.[6]

Electronics are cooled using a variety of techniques, including active cooling techniques like fans, liquid cooling, and thermoelectric coolers, as well as passive techniques like heat sinks and thermal interface materials. The size and power of the electronic item, the environment in which it operates, and the required level of cooling all influence the choice of cooling method.[8]

The rest of the paper is organized as below: Section 2 introduces the background (Methodology), Section 3 introduces our mathematical formulation. Section 4 is theresults. In Section 5 we conclude our work and future prospective.[9]

Methodology

A commercial supercomputer system using CMOS logic chips running in liquid nitrogen was developed by ETA Systems Inc. in the latter half of the 1980s [4]. The processor modules were submerged in a pool of liquid nitrogen that was kept in a cryostat vessel that was vacuum-jacketed inside the CPU cabinet. Processor circuit temperatures were kept below 90 K. Circuit speed was reported to be almost twice as fast at this temperature as it was at temperatures above ambient. The peak nucleate boiling heat flux limitations of about 12 W/cm were verified through heat transfer tests. The gaseous nitrogen created by the boiling process was recondensed using a closed-loop Stirling refrigeration device (cryogenerator). IBM started a project in 1991 to show that it was possible to package and cool a CMOS processor in a way that was appropriate for product usage [5]. The Global Server GS8900 from Fujitsu was introduced in 1999, and it used a refrigeration unit to cool a secondary coolant before supplying it to Central Processor Unit (CPU) MCMs that used liquid cooling [6]. A thorough investigation of the impact of board conduction on the predicted accuracy of Compact vThermal Models of BGA and CPGA package types was conducted by DeVoe, Jason, and Ortega, Alfonso, in 2000 [7]. They evaluated various resistance network topologies for each package style. To investigate the impact of board conductivity on CTM precision, they developed intricate (FE) isotropic board models with conductivities spanning three orders of magnitude. Using detailed board models, they compared the outcomes to those of similar fully-detailed (FE) package models. When the sensitivity to board heat conduction was great, the predictive power of the optimised topologies was substantially associated with the presence of strong local temperature gradients in the board. Board heat conduction is too low to be significant when the board has weak conductivity (k=l W/m-K). The CTMs are accurate even if the board heat flow path is dominating when the board is highly conducting (k= lOO W/m-K), as the high conductivity smoothes out local board temperature gradients. They discovered that star-shaped networks were typically only accurate to 10-15%, while optimal network topologies with shunt resistances were able to forecast the junction temperature to within 5% to 8% over a wide range of board conductivities. The emphasis was on complex two-phase flow patterns in CLTPTs for electronics cooling because of the closed loop shape and tiny tube size. To assess the friction pressure drop of the two-phase flow imposed by the available gravitational head across the loop, they employed a homogeneous two-phase flow model. To forecast the chip temperature, they employed the improved structure's boiling properties. The comparison of the observed trends with experimental data for the dielectric working fluid PF-5060 was generally in agreement through references experimented with the possibility of using the Vapotron Effect to cool electrical equipment. The issue involves a specific type of subcooled boiling that can improve heat transfer betweena non-isothermal finned surface that mimics the packaging of an electronic component and a refrigerant fluid (in this example, water) flowing across it. A relationship between the temperature trend of the water in the cavities between the fins and the cycle of events defining the phenomena has been demonstrated by experimental experiments. These findings will be beneficial to the study of more dynamic events in the future.[9]

There have been a lot of studies on cooling forelectronics chips published. Some of them have been numerically, some analytically, and the remainder has been predicted empirically. The majority of analytical solutions are brand-new methodologies that require validations. Numerical solutions include employing commercially available algorithms or alternative approaches to solve the governing equations. The process of authenticating the codes, however, is still on-going. Finally, the outcomes of the experimental research are ones that are realistically required. Therefore, in order to confirm the CFD codes and analytical conclusions, experimental results are also necessary. In this study, the governing equations will be solved using the MAC approach, and the results will be analysed.[10,11]

Mathematical Formulation

The Marker and Cell approach is used to solve the governing equations in this chapter. The full Navier- Stokes equations are solved because no assumptions are made in this case regarding the relative magnitude of the velocity components. The technique relies on the solution of a Poisson equation and finite difference discretization to determine the pressure distribution. The primitive variables u, v, w, and p are functions of x, y, z, t, and the Reynolds number Re in these approaches. In the primitive variable approach, it is implicitly assumed that the mass balance equation solves for pressure while the x-component momentum equation calculates the x- component velocity, etc. The solution described here resolves the issue that pressure is completely absent from the mass balance equation. The Navier-Stokes equations' nonlinearity is also taken into consideration. After it is finally resolved, the algorithms needed to resolve thegoverning equations are written. At the conclusion of this chapter, a single obstruction with variable dimensions is provided to my computational geometry.[17, 14]

The problem of continuity and the momentum (Navier- Stoke) equation serve as the governing equations for fluid flow. The original form of the Navier- Stokes Equations is as follows. [15,18]

Let (X Y Z) be the orthogonal components of the body force field in the Cartesian coordinate system then [16]

$$x: \quad \rho\left(\frac{\partial u^*}{\partial t^*}+\frac{\partial\left(u^{*2}\right)}{\partial x^*}+\frac{\partial\left(u^*v^*\right)}{\partial y^*}+\frac{\partial\left(u^*w^*\right)}{\partial z^*}\right)=X-\frac{\partial p^*}{\partial x^*}+\mu\left[\frac{\partial^2 u^*}{\partial x^{*2}}+\frac{\partial^2 u^*}{\partial y^{*2}}+\frac{\partial^2 u^*}{\partial z^{*2}}\right]$$

$$y: \quad \rho\left(\frac{\partial v^*}{\partial t^*}+\frac{\partial\left(u^*v^*\right)}{\partial x^*}+\frac{\partial\left(v^{*2}\right)}{\partial y^*}+\frac{\partial\left(u^*w^*\right)}{\partial z^*}\right)=Y-\frac{\partial p^*}{\partial y^*}+\mu\left[\frac{\partial^2 v^*}{\partial x^{*2}}+\frac{\partial^2 v^*}{\partial y^{*2}}+\frac{\partial^2 v^*}{\partial z^{*2}}\right]$$

$$z: \quad \rho\left(\frac{\partial w^*}{\partial t^*}+\frac{\partial\left(u^*w^*\right)}{\partial x^*}+\frac{\partial\left(v^*w^*\right)}{\partial y^*}+\frac{\partial\left(w^{*2}\right)}{\partial z^*}\right)=Z-\frac{\partial p^*}{\partial z^*}+\mu\left[\frac{\partial^2 w^*}{\partial x^{*2}}+\frac{\partial^2 w^*}{\partial y^{*2}}+\frac{\partial^2 w^*}{\partial z^{*2}}\right]$$

Figure 1 Navier stroke equation

Non-Dimensionalization

$$u=\frac{u^*}{U_\infty}, \quad v=\frac{v^*}{U_\infty}, \quad w=\frac{w^*}{U_\infty}, \quad p=\frac{p^*}{\rho U_\infty^2} \quad t=\frac{t^*}{L/U_\infty}$$

$$x=\frac{x^*}{L}, \quad y=\frac{y^*}{L}, \quad z=\frac{z^*}{L}, \quad \text{and} \quad \theta=\frac{T-T_\infty}{T_w-T_\infty} \quad \text{(for Energy equation)}$$

Figure 2 Energy Equation

Results & Discussions

Since STL doesn't preserve unit information, the imported geometry is in that format. The unit in which the model was built must be verified. The Geometry size label, which shows the model's total size in each direction, can be used to choose the unit. The default unit meter is accurate in our situation. X=0.0899m Y=0.0584m Z=0.0180m Then we create Geometry Fan after that we create box replace the created box name to fan (as per suitability) Origin [0.05 0.016 0.0115] Dimension [0.016 0.016 4e-03] Step to procedure create face group: Fan Inlet 1. Press ctrl button and bottom surface of the fan 2. Click on Geometry faces next to fan

3. Click new face group 4. Create group from 3D selection 5. Than rename that group to inlet.[12,13,25]

Create Geometry: Outlet tool The last primitive geometry to create is a box that will use to a tool extract outlet patch from outer boundaries. Select Create box Change box_1 to outlet tool Similarly set the origin & box dimension Origin [0.0845 0.0415 0] Dimension [6.5e-039e-03 8e-03]

Meshing Parameter: Fan since we are going to perform CHT (conjugate heat transfer) simulation we need to create the mesh for fluid and solid sub-domain we will specify the mesh parameter for all geometries and using the material point we will choose the subdomain that will be meshed. Hex Meshing Fan Refinement [Min 1 Max 3] Meshing Parameter: Board Refinement [Min 2 Max 3] Meshing Parameter: CPU Refinement [Min 2 Max 4]

Base Mesh: Domain Now we will define the base mesh, the box geometry determines the background of the particular of mesh domain. Steps are as followed Go to Base tab Define the box size: min[m] [0 0 0] max[m] [0.085 0.056 0.0155] Define the number of division:
Division [15 10 5] Base Mesh boundaries: First we should change boundary type of wall for all base mesh boundaries.

Solid Mesh Material Point In order to create that particular mesh in that solid region we will place the material point inside the fan geometry. The resulting mesh will remain only in this region. For this we set the value by using following procedure Go to point tab Specify location inside the CPU box Material point [0.058 0.024 5e-04]

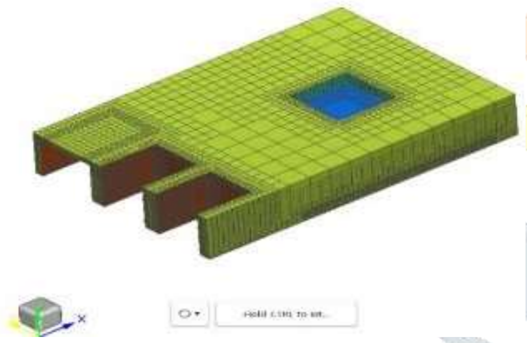


Figure 3 (Material Point)

Solid Mesh: Start Meshing

Everything is set up now for the meshing of the solid region. Then proceed with following steps Go to mesh tab Press mesh button to start meshing process. Now Solid Mesh When the meshing process is finished, the solid region mesh appears on the screen.

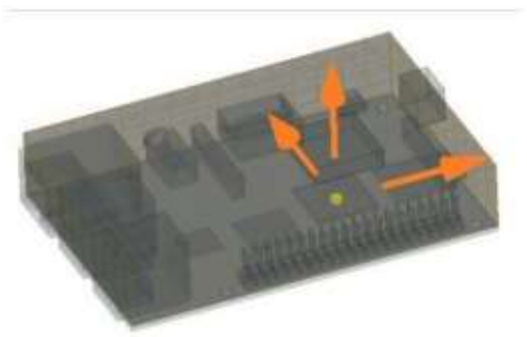


Figure 4 (Solid Mesh)

Before generating a mesh for the fluid region I must convert the current mesh into a sub region. Otherwise, it is overwritten by new mesh. Now I proceed following steps: Go to mesh panel, Expand the option list next to the default regions, Select make sub-region, Enter region name to solid then we Press Ok.[19,20]

Solid Mesh Type Expand region type option & Select solid.

Fluid Mesh Material point Once the solid region is created we can move the material point to anywhere inside the base mesh but outside the solid region. The following steps are given below Go to HexMeshing Panel Go to point tab Specify the location inside the fluidmesh Material point [0.058 0.024 5e-03] Inside the base mesh outside the Solid.

Fluid Mesh Start Meshing Everything is setup now for meshing of the fluid region. Steps to start meshing process are given below: Go to mesh tab Then process mesh button to start meshing process.[21,22]

Fluid Mesh in the graphics windows it is displayed.

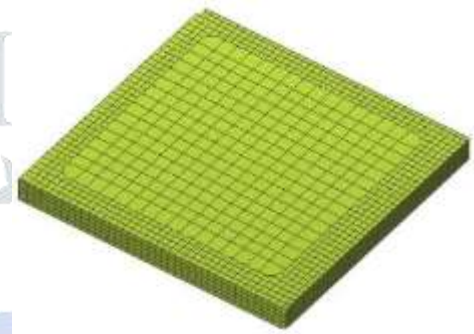


Figure 5 (fluid Mesh)

In Fluid Mesh Extract Outlet (I) In the Geometry setup, I created outlet_tool Now I will use this box to extract patch from boundary patch with the below mentioned points: Go to mesh panel then Click options button next to boundaries patch and Select extract from options.In Fluid Mesh – extract Outlet (II) Pick Outlet_tool from the list and Click extract. Fluid Mesh – Outlet Now, I have to rename the newly created boundary Change boundary name from boundaries _in_outlet_tool to outlet.

Fluid Mesh: Create sub region Now after I used the extract tool on the default mesh I can convert it into a sub region. It is important to note that extract operations are no longer available once the mesh got converted. Steps are mentioned below: Expand the listed options next to the default regions then Select make sub regions and Enter region name to fluid then we Press Ok.

Create region Interface As we already know that two mesh regions are not coupled until I create a region interface. It will be further used to define which information is exchanged between regions. Selects the CPU fluid and CPU in solid region hold ctrl key and selected both the boundaries Press create region interface.[23,24]

The steady-state equivalent of chtMultiRegionFoam is the solution chtMultiRegionSimpleFoam. For conjugate heat transfer between solid region and fluid region, the laplacian foam solution for heat conduction in solids is paired with a buoyant foam solver.

Radiation First, we'll conduct a simulation without considering radiative heat transport. All radiation model settings must be made at this time because they cannot bemodified later without restarting the simulation. Go to Radiation panel then Uncheck Enable Radiation and Set Radiation Model to Surface To Surface after Increase the Max Rays number to 3000000.[26]

Thermophysical Properties of Solid

The properties of solids and liquids must now be defined. We'll assume that the solid is formed of aluminium and the working fluid is air. Go to Thermo panel Select solid region and Click Material Database button and Select aluminium material then we will Click Apply.

Thermophysical Properties of Fluid (I) Select fluid region & Click Material Database button Select air material after that Click Apply

Thermophysical Properties of Fluid (II) we Set the Equation of State to Incompressible Perfect Gas

Turbulence For turbulence modelling, we will use Realizable k−ε model Go to Turbulence panel and then Select RANS turbulence formulation and Select Realizable k−ε model

Solution Solvers: In order to improve convergence, we will now modify the solver tolerance level of the enthalpy equation in the solid region. Go to Solution panel after that Select h (solid) tab then we Expand solver options now we put Lower solver Tolerance to 1e-08.

Solution SIMPLE we will change the settings for the SIMPLE algorithm to get better convergence. Go to the SIMPLE tab than we Increase number of Non- Orthogonal Correctors to 2

Solution – Relaxation Go to Relaxation tab and Adjust relaxation coefficients. Solution – Limits To reduce the solution's convergence space, we will now modify the temperature field bounds. Go to Limits tab and then Enable Temperature Limits Now we adjust minimum and maximum temperature to a reasonable range [Tmin 290 Tmax 600]

Cell Zones

Set the CPU volume's heat source term at this point. In this tutorial, we'll suppose that the sole component that generates heat is the CPU, which has an estimated cube volume of 200 mm. Go to Cell Zones setup Enable Source term for all cells in solid Click Add/Remove in Source Terms Select h equation Set explicit source term h Explicit [W/m3] 1250000.

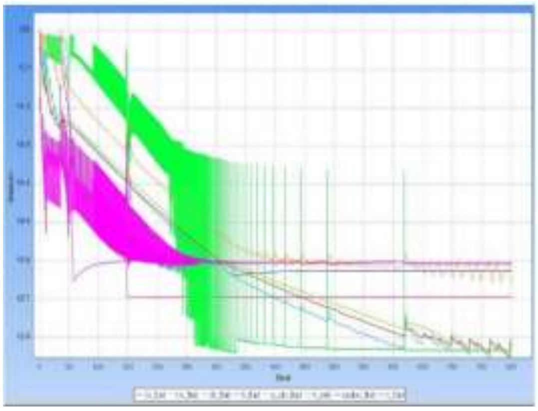
Boundary Conditions - Inlet (Flow)

The boundary conditions for the inlets and outlets will now be defined. We'll set a steady air inflow on the intake. Go to Boundary Conditions panel and then Select fan_inlet boundary and Set the Velocity Inlet character Set the inlet velocity then set theU Reference Value [m/s]0.1.

Boundary Conditions - Fluid Region (Thermal)

On the fluid side of the interface, radiation coupling will now be made possible. Click on CPU in fluid Switch to Thermal tab and Select Coupled Temperature andRadiation type.

Boundary Conditions - Solid Region (Thermal)



Next, turn on radiation coupling on the interface's solid side. Select CPU in solid and Select Coupled Temperature and Radiation type. Run Time Control Go to Run panel we Set Number of Iterations to 800 then Click Run Simulation button.

Figure 6 Residual 800

Start Postprocessing –with ParaView Launch the ParaView programme to see the outcomes. Go to Postprocessing panel then Start ParaView.

ParaView - Load Results and Select electronics_cooling.foam and Click Apply to load results then Click Last Frame to select the latest result set and After loading results they will be shown in the 3Dgraphic window

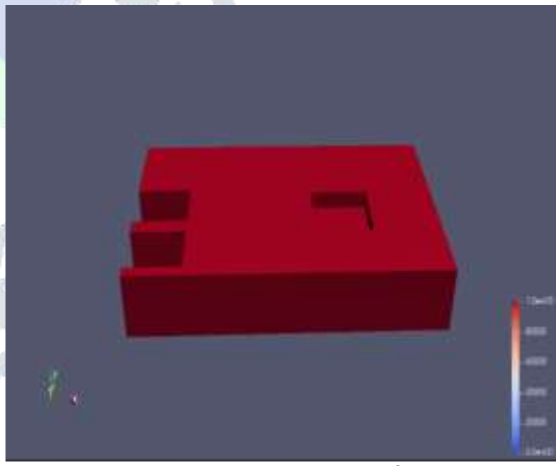


Figure 7 Electronics cooling foam

ParaView - Display Temperature Contour (I) The temperature contour will now be re-plotted on the circuit board. Since we're using the same scale, the variations in the outcomes will be more obvious. Select the followings mesh regions fluid/board fluid/cpu fluid/fan fluid/fan_inlet fluid/outlet solid/board solid/cpu solid/internalMesh (we can check Mesh Regions to select all regions and uncheck others: fluid/boundaries and fluid/internalMesh) then Click Applyand after that Select contour coloring variable to T then proceed Click Rescale to Data Range

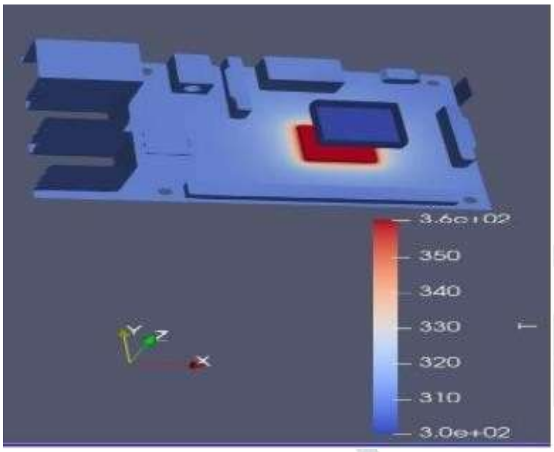


Figure 8 (Temperature contour I)

ParaView - Display Temperature Contour (II)

In the graphics window, the results are shown. Keep in mind that the domain's highest temperature is 360 K.

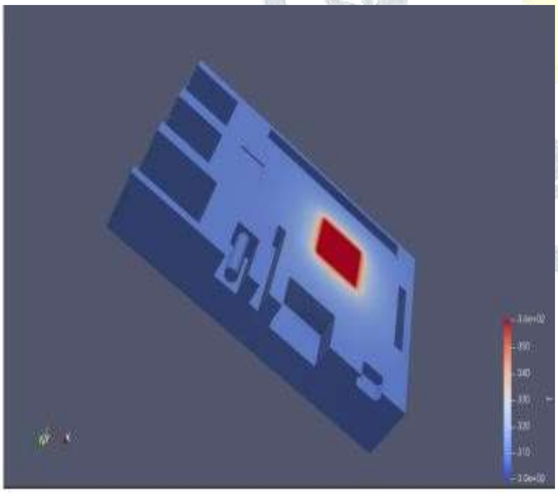


Figure 9 (Temperature contour II)

Radiation Setup : The radiation equation will now be enabled in our simulation. Close Paraview and return to SimFlow to complete this Go to Radiation panel and Check then Enable Radiation and Start Simulation with Radiation. The context is ready. We'll run the simulation and increase the number of iterations. Go to Run panel and Set Number of Iterations to 2000 then Click Continue Simulation button

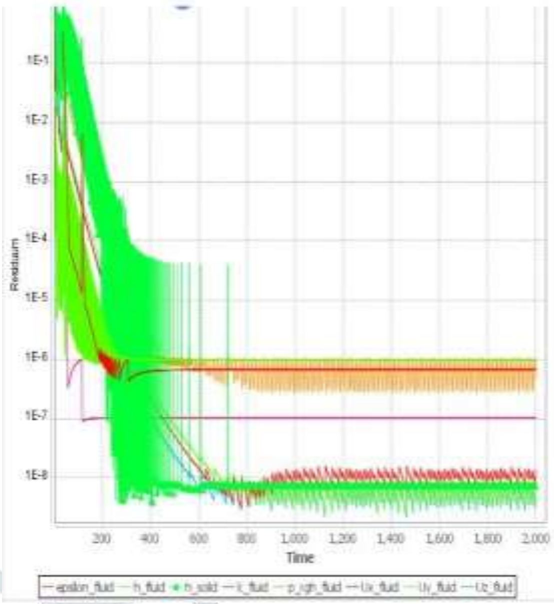


Figure 10 Residuals 2000

ParaView - Display epsilon Contour (with Radiation) (I)

The epsilon contour will now be re-plotted on the circuit board. Since you'll employ the same scale, the variations in the outcomes will be more obvious. Select the followings mesh regions fluid/board fluid/cpu fluid/fan fluid/fan_inlet fluid/outlet solid/board solid/cpu solid/internalMesh (we can check Mesh Regions to selectall regions and uncheck others: fluid/boundaries and fluid/internalMesh) and then Click Apply after apply we Select contour coloring variable to epsilon then Select Rescale to Custom Data Range and after that Set minimum and maximum value Min 300 Max 360 then again Click Rescale

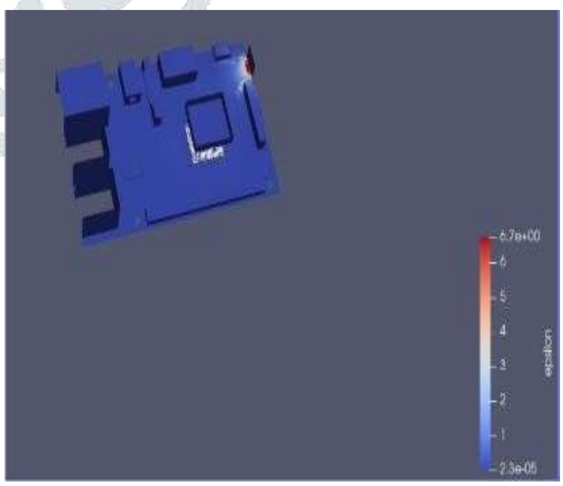


Figure 11 (epsilon contour)

ParaView - Radiative Heat Flux

Results are displayed in the graphics window.

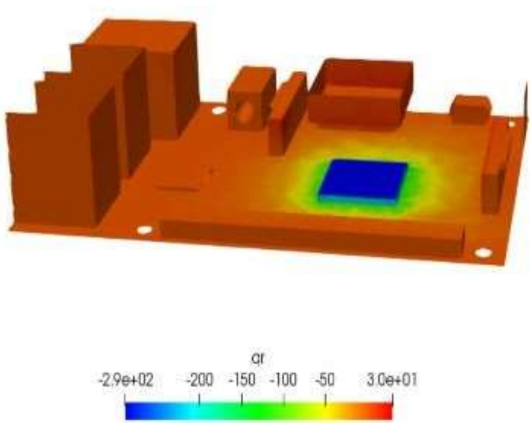


Figure 12 (radiative heat flux

)

CONCLUSIONS

For flow through a channel via an obstacle for laminar, transient flow, a computer code has been designed to solve the continuity equation, momentum equation, and energy equation with certain boundary conditions. Plotted are various pressure, temperature, and velocity profile contours as well as variations in the Nusselt number. Here, only a few of the conclusions are offered. Near the obstruction, where the pressure is more concentrated, it remains constant along the transverse direction. The Nusselt number initially rises when fresh air enters, then it falls, but when an obstacle appears, the Nusselt number rises once more, indicating a higher rate of heat transfer. After the obstruction, it lowers once again as a result of wake creation before increasing and reaching a stable state. Min/max T:360.2832 360.3344..

DILUPBiCGStab: Solving for h, Initial residual = 8.957234e-07, Final residual = 8.957234e-07, No Iterations 0 Min/max T:300 360.3323

Solving for solid region solid DICPCG: Solving for h, Initial residual = 4.901163e-09, Final residual = 4.901163e-09, No Iterations 0 DICPCG: Solving for h, Initial residual = 4.901106e-09, Final residual = 4.901106e-09, No Iterations 0 Min/max T:360.2832 360.3344 ExecutionTime = 447.08 s ClockTime = 450 s

SUGGESTIONS FOR FURTHER WORK

In this investigation, a two-dimensional scenario with a single barrier in a transitory instance was taken into consideration. There is plenty of room to expand on this work. The analysis can be done, for instance, with a problem including several chips or obstructions, either with or without a heated wall. Additionally, the identical issue can be solved in a three-dimensional domain and confirmed using the same boundary conditions using any of the market's commercially available CFD tools. Ineach of the cases, i.e., two- dimensional single chip/multi-chip and three-dimensional single chip/multi- chip, the laminar case may be extended to the turbulent case..

References

1. Davies, Mark R. D., Cole, Reena., Lohan, John., "Factors Affecting the Operational Thermal Resistance of Electronic Components", Journal of Electronic Packaging, ASME, September 2000, Vol. 122, Page 185-191.

2. Pucha, Raghuram V., Tunga,James, Krishna.,"Accelerated Thermal Guidelines for Electronic Packages in Military Avionics Thermal Environment", Journal of Electronic Packaging, ASME, June 2004, Vol.126, Page 256-264

3.Culham, J. R. , Yovanovich, M. M., Lemczyk T. F., "Thermal Characterization of Electronic Packages Using a Three-Dimensional Fourier Series Solution", Journal ofElectronic Packaging" ASME, September 2000, Vol. 122 , Page 233 -239.

4. Chen, Han-Ting., Horng, Jenn-Tsong., Chen, Po-Li., Hung, Ying-Huei., " Optimal Design for PPF Heat Sinks in Electronics Cooling Applications", Journal ofElectronic Packaging, ASME, December 2004, Vol. 126,Page 410-422.

5. Zhao, C.Y. , Lu, T.J., "Analysis of Micro Channel Heat Sinks For Electronics Cooling", International Journal of Heat and Mass Transfer, 2002, Vol.45, Page 4857-4869.

6.Egan, Eric., Amon Cristina H., "Thermal Management Strategies fir Embedded Electronic Components ofWearable Computers", Journal of Electronic Packaging, June 2000, Vol.122, Page 98- 106.

7. Hung, T. C., Wangi S. K., & Peter F., "Simulation of Passively Enhanced Conjugate Heat Transfer Across an Array of Volumetric Heat Sources" Communications in Numerical Methods in Engineering, John Wiley & Sons, Ltd. Vol.13, 1997,Page 855-866.

8.Mukhopadhyay, A., Biswas, G., Sundararajan, T., "Numerical investigation of confined wakes behind a square cylinder in a channel", Int. J. Numerical Methods in Fluids, Vol. 14, pp 1473-1484, 1992.

9. Gauch, Paul., and Xu, Weiran., "Modelling Phase Change Material in Electronics Using CFD- A Case Study", International Conference on High DensityInterConnect and Systems Packaging, 2000.

10. Kitamura, Yoji., Ishizuka, Masaru., "Chimney Effect on Natural Cooling of Electronic Equipment Under Inclination.", Journal of Electronic Packaging, December2004, Vol.126., Page 423-429

11. Bhopte, Siddharth.,. Alshuqairi, Musa S., Agonafer, Dereje., Ahmed, Gamal Refai., "Mixed Convection of Impinging Air Cooling Over Heat Sink in Telecom System Application", Journal of Electronic Packaging, ASME, December 2004, Vol. 126, Page 519-523.

12. Cheng.Y.P , Lee.T.S & Low.H.T, "NumericalAnalysis of Mixed Convection in Three Dimensional Rectangular Channel with Flush Mounted Heat Sources Based on Field Synergy Principle", International Journal for numerical methods in fluids(in press)(2006) John Wiley & Sons Ltd.

13. Haider, S. I.,. Joshi, Yogendra, K., Nakayama, Wataru., "A Natural Circulation Model of the ClosedLoop Two Phase Thermo Syphon for Electronics Cooling" Journal of Heat Transfer, ASME, October,2002

14. Gima, Satoru., Nagata, Takashi., Zhang, Xing., and Fujii, Motoo., " Experimental Study on CPU Cooling System of Closed Loop Two Phase Thermosyphon", Heat Transfer- Asian Research 34(3) - 2005 Wiley Periodicals

15. Rhee, Jinny., Moffat, Robert J., "Experimental Estimate of the Continuous OneDimensional Kernel Function in a Rectangular Duct with Forced Convection", Journal of Heat Transfer, ASME, August 2006, Vol-128, Page 811-818.

16. Orlanski, I., "A Simple Boundary Condition for Unbounded Flows", Journal of Computational Physics, Vol. 21, Page 251-269, 1976.

17. Bergles, A. E. "The evolution of cooling technology for electrical, electronic, and microelectronicequipment," ASME HTD, vol. 57, pp. 1–9, 1986.

18. Schwall, R. E., and Harris, W. S., "Packaging and cooling of low temperature electronics," in Advances in Cryogenic Engineering. New York: Plenum Press, 1991, pp. 587–596.

19. Roy, A., Bandyopadhyay, G., "Numerical Investigation of Confined Flow past a Square Cylinder Placed in a Channel." IE (I), Vol 85, November 2004.

20. Dhiman, A.K., "Chhabra, R.P., Eswaran, V., "Flow and Heat Transfer Across a Confined Square Cylinder in the Steady Flow Regime: Effect of Pectet Number", International Journal of Heat and Mass Transfer, 48 (2005), Page 4598-4614.

21. Bebnia, Masud., Nakayama, Wataru., and Wan, Jeffrey., "CFD Simulations of Heat Transfer from a Heated Module in an Air Stream: Comparison with Experiments and a Parametric Study" IEEE 1998, Inter Society Conference on Thermal Phenomena.

22. . Grimes, Ronan., Davies, Mark., "Air Flow and Heat Transfer in Fan Cooled Electronic Systems", Journal of Electronic Packaging, ASME, Vol.126, March 2004, Page 124-134.

23. Tae Ho Ji a, Seo Young Kim b, and Jae Min Hyuna, “Heat Dissipation from a Heated Square Cylinder in Osvillating Channel Flow”, IEEE 2006.

24. Harlow, F.H and Welch, J.E., “Numerical Calculation of Time dependent Viscous incompressible flow of Fluid with free surfaces”, The Phys of Fluid, Vol.8, pp2182 2188, 1965.

25.https://help.sim-flow.com/tutorials/electronics- cooling.

26. Rodgers, Peter J., Eveloy, Valerie C., Davies, Mark
R. D., “ An Experimental Assessment of Numerical Predictive Accuracy for Electronic Component Heat Transfer in Forced Convection-Part 2, Results and Discussions”, Journal of Electronic Packaging, ASME, March 2003, Vol-125, Page 76-83.