MULTI-VARIABLE DESIGN OPTIMIZATION OF A CONTEMPORARY COLD PLATE FOR A FIXED PUMPING POWER FOR MINIMIZING THE THERMAL RESISTANCE

by

PRATIK KEKRE

Presented to the Faculty of the Graduate School of
The University of Texas at Arlington in Partial Fulfillment
of the Requirements
for the Degree of

MASTER OF SCIENCE IN MECHANICAL ENGINEERING

THE UNIVERSITY OF TEXAS AT ARLINGTON
August 2014

Acknowledgements

I take this opportunity to thank firstly my supervising advisor, Dr. Dereje Agonafer, for guiding me, supporting me and believing in me. This helped me in itself a lot to carry forward the learning process.

Secondly, my committee members, Dr. Haji Sheikh and Dr. Kent Lawrence, for taking out time from their busy schedule and provide me with useful and helpful comments and suggestions. My PhD mentor, John Fernandes for his invaluable time and his timely inputs. Without his support this project would not have been a success. Sally Thompson, who constantly motivated and encouraged me to keep moving forward during my difficult and hard times.

Thirdly, my colleagues at EMNSPC and my friends who have been there with me throughout this effort and have given undivided support and motivation to achieve it.

Lastly, I would like to thank my parents and my family. Without their support and their sacrifices, this dream would not be a reality.

.

April 08, 2014

Abstract

MULTI-VARIABLE DESIGN OPTIMIZATION OF A CONTEMPORARY COLD PLATE FOR A FIXED PUMPING POWER FOR MINIMIZING THE THERMAL RESISTANCE

Pratik Kekre, MS

The University of Texas at Arlington, 2014

Supervising Professor: Dereje Agonafer

Due to reliability concerns, thermal management of microprocessors has and continues to become a major challenge in electronic packaging. High circuit densities in modern integrated circuit semiconductor devices require the heat generated by their operation be efficiently removed, in order to maintain the temperatures of the device within limits which will consequently follow the design guideline for operating parameters.

Air-cooling is still the preferred method of cooling electronic systems and especially in terms of cost. Air cooling, however, is starting to reach its limits for some of the higher end electronic systems and as such there is a need to investigate the application of liquid cooling for cooling high end servers. Also, it has been proved that airflow through circuit boards have inherently low heat transfer coefficients and large pressure drops, and hence requires large heat transfer areas with considerable amount of flow sections. As a result, liquid cooling has grown in prominence as a method for cooling high density interconnect (HDI) devices. Water cooling in particular has multiple advantages over air cooling as it has greater heat carrying capacity. Cold plates enable the use of water with its excellent cooling capability and proven reliability. Previously, optimization work has been done for a classic formed tube cold plate by varying 2 parameters for a fixed pumping power. In this paper we address optimizing the design of a contemporary cold plate for a fixed pumping power by varying multiple design parameters such as the contact width, radius of the curvature, height of the tube. A design of experiment (DOE) on Computational Fluid Dynamics (CFD) is created and deployed such that it maximizes the co-efficient of performance of the cold plate. This would serve as a guideline in the future for design of high performance cold plates that would be employed for thermal management of high power single or multichip modules.

Table of Contents

List of Figures

List of Tables

Chapter 1

Introduction

> "Scientists discover the world that exists; engineers create the world that never was." — Theodore Von Karman.

Over the past few decades there has been a steep rise in the power consumption of high end electronic equipment, which in turn has led to more dissipation of heat, this places an enormous load on the performance of the cooling systems. This perspective has made it difficult to rely on air-cooled cooling systems. Air cooling basically requires extensive amount of pumping power to cool high end electronic equipment due to high inlet air temperature. Hence there is a need to develop techniques that would suffice the thermal management requirement in the current and in the future, such as, water cooled cold plates.

Liquid cooling is, in itself, is not yet a matured technology. The advantages of water cooling over air cooling include water's higher specific heat capacity and thermal conductivity. This project highlights the technology of liquid cooled heat sink deployed for the thermal management of high end servers.

Literature Survey:

Liquid cooling of electronics is not a new technology. The need to further increase packaging density and reduce signal delay between communicating circuits led to the development of multi-chip modules beginning in the late 1970s. The heat flux associated with bipolar circuit technologies steadily increased from the very beginning and really took off in the 1980s.[1] IBM had determined that the most effective way to manage chip temperatures in these systems was through the use of indirect water-cooling.[2] Several other mainframe manufacturers also came to the same conclusion. The decision to switch from bipolar to Complementary Metal Oxide Semiconductor (CMOS) based circuit technology in the early 1990s led to a significant reduction in power dissipation and a return to totally air-cooled machines. However, this was but a brief respite as power and packaging density rapidly increased, matching then exceeding the performance of the earlier bipolar machines. These increased packaging densities and power levels have resulted in unprecedented cooling demands at the package, system and data center levels necessitating a return to water-cooling.[3]

Then back in 2008 IBM reintroduced water cooling technology into its high performance computing platform, the Power 575 Supercomputing node/system. Water cooled cold plates were used to cool the processor modules which represented about half of the total system (rack) heat load. An air-to-liquid heat exchanger was also mounted in the rear door of the rack to remove a significant fraction of the other half of the rack heat load; the heat load to air. Water cooling enabled a compute node with 34% greater performance (Flops), resulted in a processor temperature 20-30 ^{o}C lower than that typically provided with air cooling, and reduced the power consumed in the data center to transfer the IT heat to the outside ambient by as much as 45%.

Thermal Management of high end electronic devices

Heat generated by electronic devices must be dissipated to improve reliability and prevent failure. Techniques used till date for heat dissipation include heat sinks and fans for air cooling, and cold plates for liquid cooling.

Air Cooling Methodologies:

Heat Sinks:

In electronic systems, a heat sink is a passive heat exchanger that cools a device by dissipating heat into the surrounding medium. In computers, heat sinks are used to cool central processing units (CPU) or graphic processing units (GPU). Heat sinks are used with high-power semiconductor devices such as power transistors and optoelectronics such as lasers and light emitting diodes (LEDs), where the heat dissipation ability of the basic device is insufficient to moderate its temperature.

Working Principle:

Heat sinks work on the principle of Fourier's Law of Conduction which when simplified to a one-dimensional form in the *x*-direction, states that when there is a temperature gradient in a body, heat will be transferred from the higher temperature region to the lower temperature region. The rate at which heat is transferred by conduction, q_k, is proportional to the product of the temperature gradient and the cross-sectional area through which heat is transferred. [4]

$$q_k = -kA\frac{dT}{dx}$$

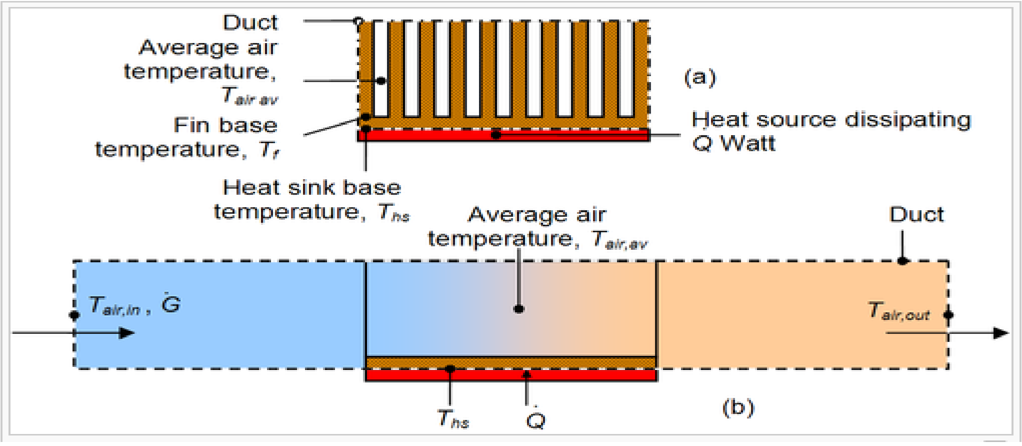


Figure 1: Principle of Conduction

Material:

The materials commonly used are aluminum alloys along with copper, diamond and other composite materials. Aluminum alloy 1050A has one of the higher thermal conductivity values at 229 W/mK but is mechanically soft. Aluminum alloys 6061 and 6063 are commonly used, with thermal conductivity values of 166 and 201 W/mK, respectively. The values depend on the alloy.

Copper has excellent heat sink properties in terms of its thermal conductivity, corrosion resistance, bio fouling resistance, and antimicrobial resistance. Copper has around twice the thermal conductivity of aluminum and faster, more efficient heat absorption. Copper is three times as dense as and more expensive than aluminum. Copper heat sinks are machined and skived. Another method of manufacture is to solder the fins into the heat sink base. Aluminum can be extruded, but copper cannot.

Diamond is another heat sink material, and its thermal conductivity of 2000 W/mK that exceeds copper five-fold. In contrast to metals, where heat is conducted by delocalized electrons, lattice vibrations are responsible for diamond's very high thermal conductivity.

Thermal Interface Materials:

Thermal interface materials (TIM) are a common way to overcome thermal contact limitations such as gaps, caused by the roughness of the surface, voids, created by the misalignment of the surface. Properly applied thermal interface materials displace the air that is present in the gaps between the two objects with a material that has a much-higher thermal conductivity. Air has a thermal conductivity of 0.022 W/mK, while TIMs have conductivities of 0.3 W/mK and higher. Selection of a TIM is based on three parameters: the interface gap which the TIM must fill, the contact pressure, and the electrical resistivity of the TIM. The contact pressure is the pressure applied to the interface between the two materials. The selection does not include the cost of the material. Electrical resistivity may, or may not, be important, depending upon electrical design details. [5]

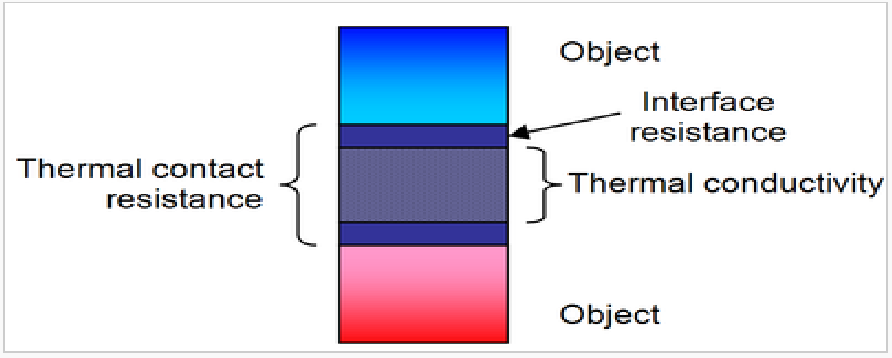
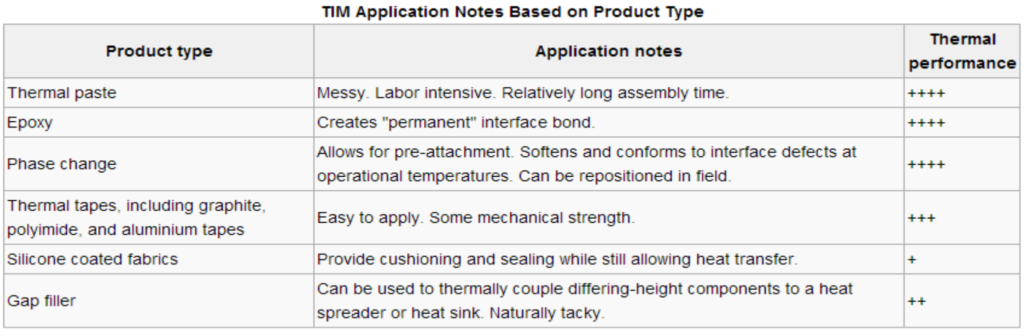


Figure 2: Thermal Interface Materials



TIM Application Notes Based on Product Type

Product type	Application notes	Thermal performance
Thermal paste	Messy. Labor intensive. Relatively long assembly time.	++++
Epoxy	Creates "permanent" interface bond.	++++
Phase change	Allows for pre-attachment. Softens and conforms to interface defects at operational temperatures. Can be repositioned in field.	++++
Thermal tapes, including graphite, polyimide, and aluminium tapes	Easy to apply. Some mechanical strength.	+++
Silicone coated fabrics	Provide cushioning and sealing while still allowing heat transfer.	+
Gap filler	Can be used to thermally couple differing-height components to a heat spreader or heat sink. Naturally tacky.	++

Figure 3: Applications of Thermal interface materials

Fans:

While in earlier personal computers it was possible to cool most components using natural convection (passive cooling), many modern components require more effective active cooling. To cool these components, fans are used to move heated air away from the components and draw cooler air over them. Fans attached to components are usually used in combination with a heat sink to increase the area of heated surface in contact with the air, thereby improving the efficiency of cooling.

1. Cooling Fan Applications:
2. Case Mount
3. CPU fan
4. Graphics Card fan
5. Chipset Fan

Liquid Cooling Methodologies:

Cold Plates:

Cooling hot computer components with various fluids has been in use since at least as far back as the development of Cray-2 in 1982, using Fluorinet. Through the 1990s, water cooling for home PCs slowly gained recognition amongst enthusiasts, but it started to become noticeably more prevalent after the introduction of AMD's hot-running Athlon processor in mid 2000. As of 2011, there are several manufacturers of water cooling components and kits, and some custom computer retailers include various setups of water cooling for their high performance systems. Water cooling usually uses a CPU water block, a water pump, and a heat exchanger (usually a radiator with a fan attached). Water cooling can allow quieter (potentially fan less) operation, or improved processor speeds, or a balance of both. Less commonly, GPU’s, North bridges, South bridges, hard disk drives, memory, voltage regulator modules (VRMs), and even power supplies can be water-cooled.

Immersion Cooling:

An uncommon practice is to submerge the computer's components in a thermally, but not electrically, conductive liquid. Although rarely used for the cooling of computers, liquid submersion is a routine method of cooling large power distribution components such as transformers. Personal computers cooled in this manner do not generally require fans or pumps, and may be cooled exclusively by passive heat exchange between the computer's parts, the cooling fluid and the ambient air. Extreme component density supercomputers such as the Cray-2 and Cray T90 used additional liquid-to-chilled liquid heat exchangers for heat removal.

The liquid used must have sufficiently low electrical conductivity not to interfere with the normal operation of the computer. If the liquid is somewhat electrically conductive, it may be necessary to insulate certain parts of components susceptible to electromagnetic interference, such as the CPU.[7] For these reasons, it is preferred that the liquid be dielectric.

A wide variety of liquids exist for this purpose, the most suitable being transformer oils and other specialty electrical cooling oils such as 3M Fluorinert. Non-purpose oils, including cooking, motor and silicone oils, have been successfully used for cooling personal computers.

Evaporation can pose a problem, and the liquid may require either be regularly refilling or sealing inside the computer's enclosure.

Chapter 2

Motivation and Modelling

Motivation

Air cooling is basically energy intensive. Exorbitant amounts of pumping power can be consumed in cooling extremely high power server nodes, especially when the inlet air temperatures are high. Thus, the need to cool current and future high heat load high heat flux electronics mandates the development of extremely aggressive and highly energy efficient thermal management techniques, such as liquid cooling using cold plate devices.

This project proposes a solution to implement a cold plate having copper tube brazed in an aluminum model body. The idea of cold plates has being put to use in the industry for the past several years. This project aims at providing an optimization methodology for minimizing the thermal resistance of a contemporary cold plate by fixing the pumping power.

The idea is basically to optimize a cold plate design between two reference designs. The two designs happen to be the extreme cases for the optimization that was carried.

Reference Design 1*:*

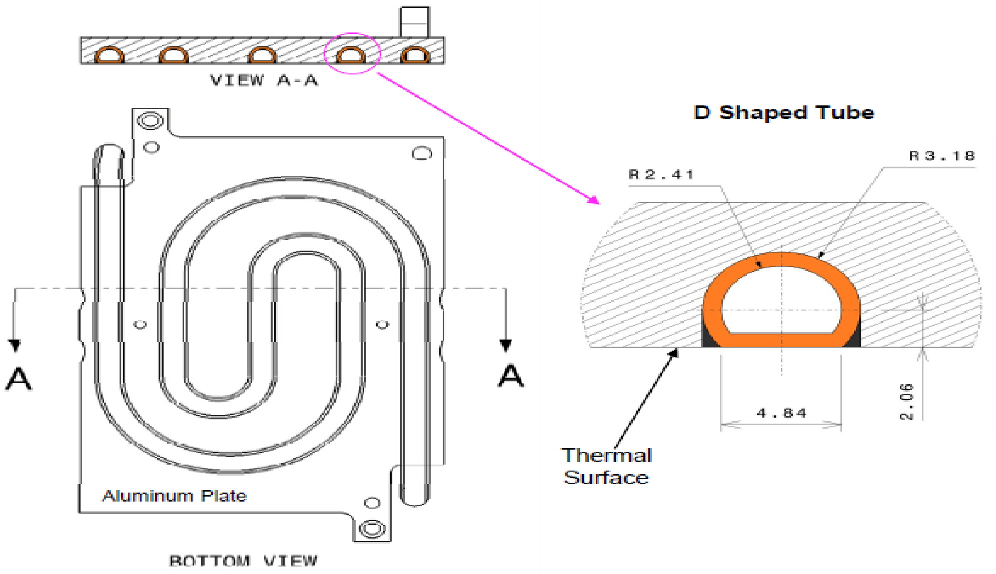


Figure 4: Reference cold plate for a QCM

The assembly of the cold plate as shown in figure is a copper tube brazed in aluminum plates. The plate is designed for a Quad chip module (IBM 775 Super compute Module) with a dissipating heat capacity of 150 W. Module encapsulation is accomplished with a single multifunctional copper alloy lid. Copper is chosen for efficient heat transfer. A silicone elastomeric thermal interface material thermally joins each processor chip to the copper alloy lid (TIM1) while an indium pad serves as the thermal interface material between the module lid and an external cold plate (TIM2). The lid also functions to provide Land-Grid Array (LGA) connector alignment, module-to-PCB alignment, cold-plate alignment and attachment as well as surface for module part number, serial number and traceability. The lid is adhered to the HPGC carrier with an elastomeric seal. The entire module assembly including water-cooled cold plate is electrically and mechanically interconnected to the PCB through the LGA interconnection system.[6]

Reference Design 2*:*

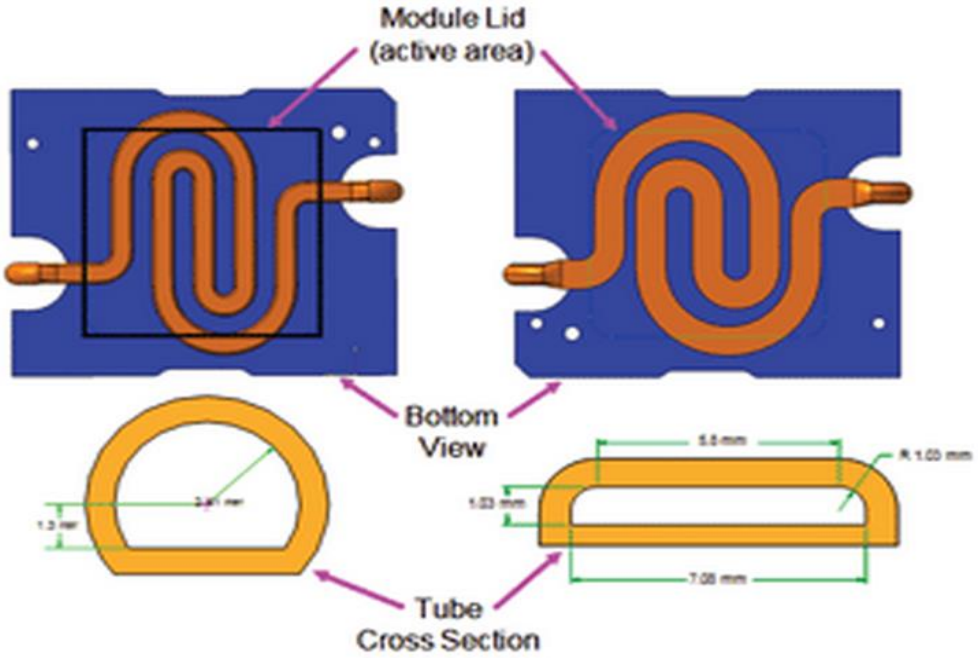


Figure 5: Reference cold plate for a SCM

The figure above is a consideration of a low-cost copper tube in aluminum plate for single chip modules. The Design aspects being similar to the cold plate designed for the IBM 775 Super compute Module.[7]

The Model:

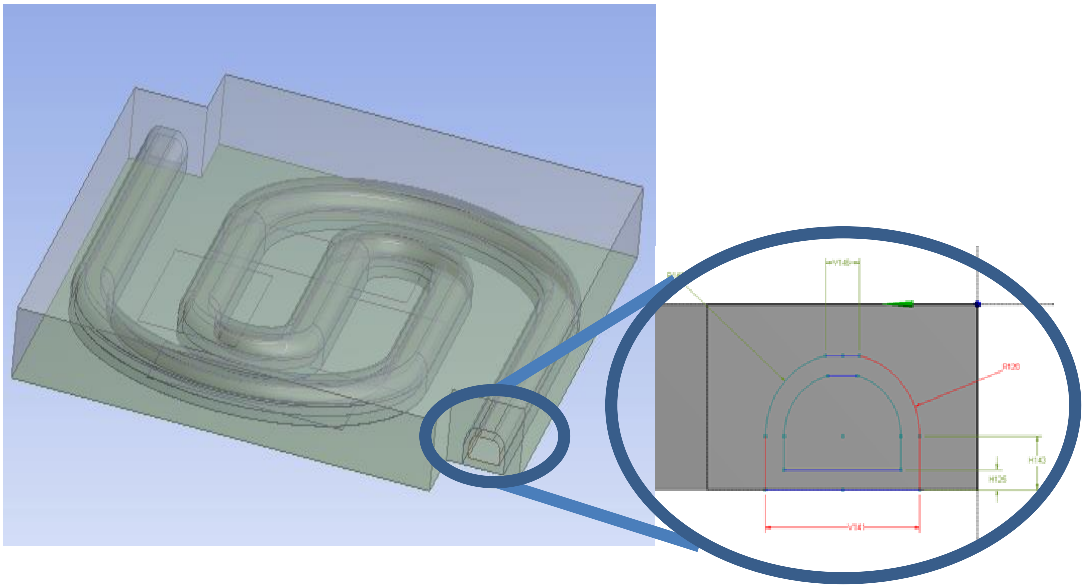


Figure 6: CAD model for the cold plate

The model is created in Creo 2.0. The model created is an aluminum plate having a copper tube brazed in it. The objective of this project is to optimize the dimensions of the copper tube such that it provides minimum thermal resistance to the heat dissipation of the module.

Parameters (Design Variables):

The design variables are the quantities that may be varied in the system in order to satisfy the objective. Hence, during the design process, attention is focused on these parameters which when varied determine the behavior of the thermal system so that the objective is achieved. It is therefore important to focus on the main design variables in the problem because the complexity of the design variable is a function of the number of variables.

- Independent Variables:

The channel width, the channel height and the channel radius are the independent variable for this design optimization.

- Dependent Variables:

The tube thickness is a fixed parameter. Also the inner cross section of the tube is dependent on the independent variables and the tube thickness.

Chapter 3

Mesh and Mesh Sensitivity Analysis

Meshing Fundamentals

Purpose of Meshing:

The model is meshed for solving the equations at a cell/nodal locations of the model. The domain is required to be divided into discrete cells.

Requirements:

- Efficiency and Accuracy:

The cells at the area of interest need to be refined to fine cells for high solution gradients and fine geometric detail. The mesh can be coarser (larger cells) elsewhere.

- Quality:

The solution accuracy and stability deteriorates as the mesh gets coarser elsewhere. So it is important to have a finer mesh at the location of interest.

Meshing Process in ANSYS:

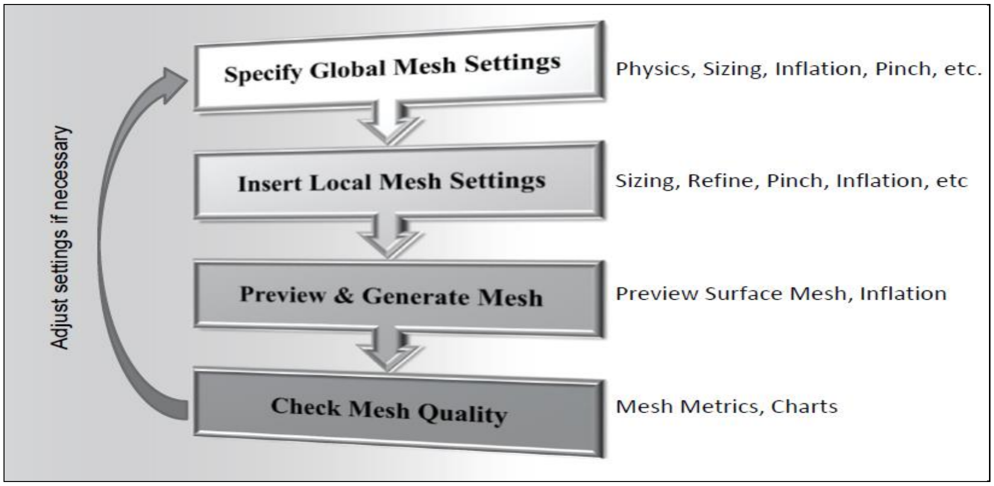


Figure 7: Mesh Methodology in ANSYS

Assembly Meshing:

Assembly type of meshing was used to mesh this particular model as the model was not sweep able. Assembly meshing typically operates on parts, multi bodies etc. It also tolerated overlapping bodies. It creates conformal mesh across the parts in contact, hence eliminating the need for multi body part generation in the associated CAD software. It also has the ability to form virtual bodies for ‘fluid flow’ from closed set of bodies. This eliminated the use of a fill or Boolean operation in CAD. Assembly meshing is patch independent and so eliminates the need for pinch control. Mesh elements size is driven by size functions. Assembly meshing is divided into 2 sub-types:

- Cut Cell
- Tetrahedrons

Model Meshing:

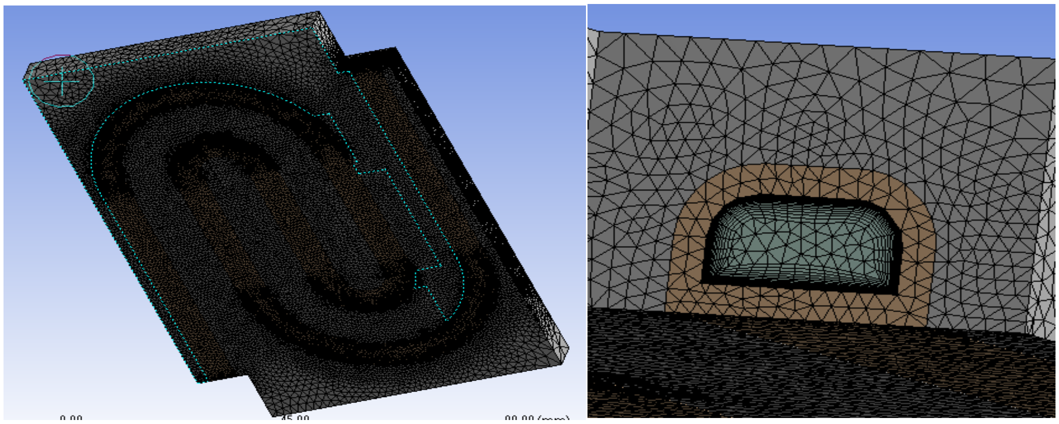


Figure 8: (a)Mesh generated on the model (b) inflation layers on the fluid region

Tetrahedron type of Assembly meshing is deployed to mesh the model. Tetrahedron mesh generates a patch independent tetra mesh with automatic defeaturing. It is compatible with inflation (Quality Check). The tetrahedron mesh works in the following manner:

- Generate Cut cells
- Delete volume mesh

- Triangulate surface mesh
- Fill with tetra mesh

Mesh Sensitivity Check:

In order to have a sensitivity check on the mesh creation, inflation layers are created on the fluid boundary. Inflation layers are deployed to capture the boundary layer effects (generally in case of turbulent flow).

Mesh Sensitivity Analysis:

The number of inflation layers are varied and the corresponding output parameters are recorded for the sensitivity analysis.

Table 1: Mesh Sensitivity Analysis

Inflation layers	No. of elements	Max Skewness	Ppump	Rcp
5	2368554	0.899023554	0.007973	0.085614
12	2741207	0.897867589	0.008047	0.086597
15	2919760	0.899376545	0.0081	0.088244
18	3098582	0.899389234	0.008172	0.069197
21	3282880	0.899197656	0.008311	0.090213
25	3528653	0.898706392	0.008663	0.084833
27	3651433	0.899004395	0.009021	0.089904

Chapter 4

FLUENT: Setup and Solution

Heat Transfer:

Heat is defined in physics as the transfer of thermal energy across a well-defined boundary around a thermodynamic system. And heat transfer is a process function (or path function), as opposed to functions of state; therefore, the amount of heat transferred in a thermodynamic process that changes the state of a system depends on how that process occurs, and not only the net difference between the initial and final states of the process. Thermodynamic and mechanical heat transfer is calculated with the heat transfer coefficient, the proportionality between the heat flux and the thermodynamic driving force for the flow of heat. In engineering contexts, the term heat is taken as synonymous to thermal energy. This usage has its origin in the historical interpretation of heat as a fluid (caloric) that can be transferred by various causes,[4] and that is also common in the language of laymen and everyday life. The transport equations for thermal energy (Fourier's law), mechanical momentum (Newton's law for fluids), and mass transfer (Fick's laws of diffusion) are similar,[5][6] and analogies among these three transport processes have been developed to facilitate prediction of conversion from any one to the others.[6] Thermal engineering concerns the generation, use, conversion, and exchange of heat transfer. As such, heat transfer is involved in almost every sector of the economy.[7] Heat transfer is classified into various mechanisms, such as thermal conduction, thermal convection, thermal radiation, and transfer of energy by phase changes.

Modes of Heat Transfer:

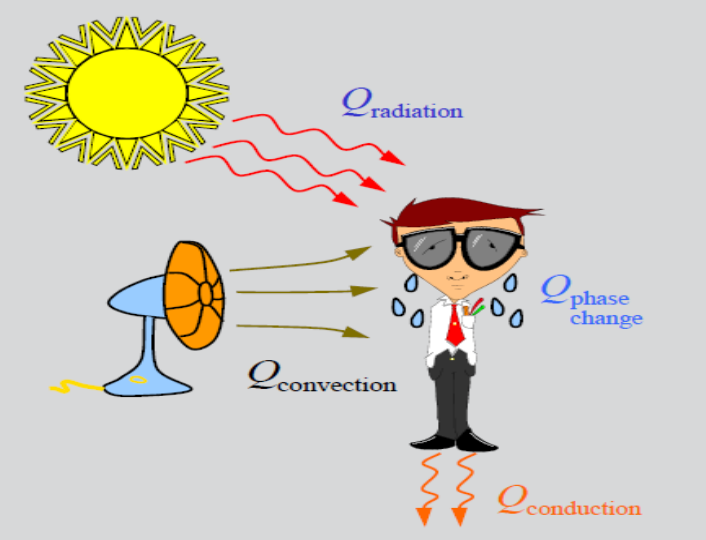


Figure 9: Modes of Heat transfer

Conduction:

Conduction is the most significant means of heat transfer within a solid or between solid objects in thermal contact. Conduction is greater in solids because the network of relatively close fixed spatial relationships between atoms helps to transfer energy between them by vibration. Fluids (and especially gases) are less conductive. This is due to the large distance between atoms in a gas; fewer collisions between atoms means less conduction. Conductivity of gases increases with temperature. Conductivity increases with increasing pressure from vacuum up to a critical point that the density of the gas is such that molecules of the gas may be expected to collide with each other before they transfer heat from one surface to another. After this point conductivity increases only slightly with increasing pressure and density.

Metals (e.g., copper, platinum, gold, etc.) are usually good conductors of thermal energy. This is due to the way that metals bond chemically. The electron fluid of a conductive metallic solid conducts most of the heat flux through the solid. Electrons also conduct electric current through conductive solids, and the thermal and electrical conductivities of most metals have about the same ratio. A good electrical conductor, such as copper, also conducts heat well.

To quantify the ease with which a particular medium conducts, engineers employ the thermal conductivity, also known as the conductivity constant or conduction coefficient, k. In thermal conductivity k is defined as "the quantity of heat, Q, transmitted in time (t) through a thickness (L), in a direction normal to a surface of area (A), due to a temperature difference (ΔT) [...]." Thermal conductivity is a material property that is primarily dependent on the medium's phase, temperature, density, and molecular bonding. Thermal effusivity is a quantity derived from conductivity, which is a measure of its ability to exchange thermal energy with its surroundings.

Forms of Conduction:

1. Steady State Conduction
2. Transient Conduction
3. Relativistic Conduction
4. Quantum Conduction

Fourier’s Law of Conduction:

The law of heat conduction, also known as Fourier's law, states that the time rate of heat transfer through a material is proportional to the negative gradient in the temperature and to the area, at right angles to that gradient, through which the heat flows. Mathematically,

$$\frac{Q}{A} = q = -\underset{\uparrow\,\text{Thermal conductivity (not necessarily constant)}}{k}\,\nabla T = -k\left(\frac{\partial T}{\partial x} + \frac{\partial T}{\partial y} + \frac{\partial T}{\partial z}\right)$$

This leads us to the concept of Thermal Resistance:

$$T_{\text{hot}} - T_{\text{cold}} = R\,Q \qquad\qquad R = \frac{t}{k\,A}$$

Convection:

Convection is the transfer of thermal energy from one place to another by the movement of fluids. Although often discussed as a distinct method of heat transfer, convection describes the combined effects of conduction and fluid flow or mass exchange. Two types of convective heat transfer may be distinguished:

- <u>Free or natural convection:</u> When fluid motion is caused by buoyancy forces that result from the density variations due to variations of thermal temperature in the fluid. In the absence of an external source, when the fluid is in contact with a hot surface, its molecules separate and scatter, causing the fluid to be less dense. As a consequence, the fluid is displaced while the cooler fluid gets denser and the fluid sinks. Thus, the hotter volume transfers heat towards the cooler volume of that fluid.[2] Familiar examples are the upward flow of air due to a fire or hot object and the circulation of water in a pot that is heated from below.
- <u>Forced convection:</u> when a fluid is forced to flow over the surface by an external source such as fans, by stirring, and pumps, creating an artificially induced convection current.[3]

Internal and external flow can also classify convection. Internal flow occurs when a fluid is enclosed by a solid boundary such when flowing through a pipe. An external flow occurs when a fluid extends indefinitely without encountering a solid surface. Both of these types of convection, either natural or forced, can be internal or external because they are independent of each other.

Newton’s Law of Cooling:

Newton's law, which requires a constant heat transfer coefficient, states that *the rate of heat loss of a body is proportional to the difference in temperatures between the body and its surroundings*. The rate of heat transfer in such circumstances is derived below.

Newton's cooling law is a solution of the differential equation given by Fourier’s Law:

$$\frac{dQ}{dt} = h \cdot A \cdot (T(t) - T_{\text{env}}) = h \cdot A \Delta T(t)$$

Where,

Q is the thermal energy in joules,

h is the heat transfer coefficient (assumed independent of T here) (W/m^2 K),

A is the surface area of the heat being transferred (m^2),

T is the temperature of the object's surface and interior (since these are the same in this approximation),

T_{env} is the temperature of the environment; i.e. the temperature suitably far from the surface,

$\Delta T(t) = T(t) - T_{\text{env}}$ is the time-dependent thermal gradient between environment and object.

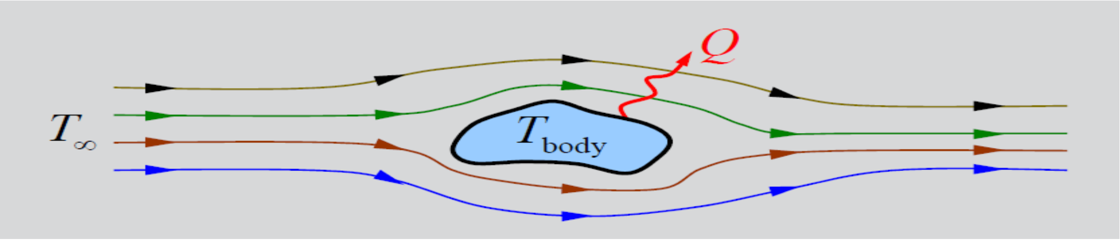


Figure 10: Convection principle

Radiation:

Thermal Radiation occurs through a vacuum or any transparent medium (solid or fluid). It is the transfer of energy by means of photons in electromagnetic waves governed by the same laws. Earth's radiation balance depends on the incoming and the outgoing thermal radiation. Anthropogenic perturbations in the climate system, are responsible for a positive radiative forcing which reduces the net long wave radiation loss out to Space. Thermal radiation is energy emitted by matter as electromagnetic waves, due to the pool of thermal energy in all matter with a temperature above absolute zero. Thermal radiation propagates without the presence of matter through the vacuum of space.

Thermal radiation is a direct result of the random movements of atoms and molecules in matter. Since these atoms and molecules are composed of charged particles (protons and electrons), their movement results in the emission of electromagnetic radiation, which carries energy away from the surface.

The Stefan-Boltzmann equation, which describes the rate of transfer of radiant energy, is as follows for an object in a vacuum:

$$Q = \epsilon\sigma T^4$$

For radiative transfer between two objects, the equation is as follows:

$$Q = \epsilon\sigma(T_a^4 - T_b^4)$$

Advection:

By transferring matter, energy, including thermal energy, is moved by the physical transfer of a hot or cold object from one place to another. This can be as simple as placing hot water in a bottle and heating a bed, or the movement of an iceberg in changing ocean currents. A practical example is thermal hydraulics. This can be described by the formula:

$$Q = v \cdot \rho \cdot c_p \cdot \Delta T$$

Where Q is heat flux (W/m²), ρ is density (kg/m³), c_p is heat capacity at constant pressure (J/(kg*K)), ΔT is the change in temperature (K), v is velocity (m/s).

Boundary Layer Flow: (Thermal and Viscous Boundary Layer)

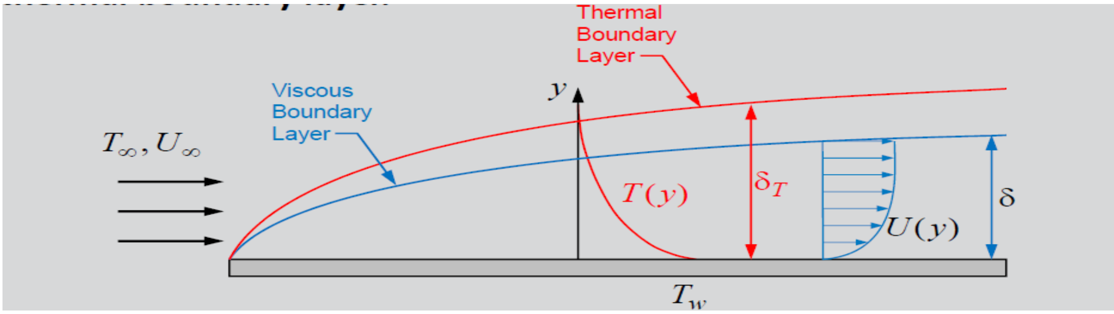


Figure 11: Boundary layer effects

In physics and fluid mechanics, a boundary layer is the layer of fluid in the immediate vicinity of a bounding surface where the effects of viscosity are significant. The thickness of the velocity boundary layer is normally defined as the distance from the solid body at which the viscous flow velocity is 99% of the free stream velocity (the surface velocity of an inviscid flow). Displacement Thickness is an alternative definition stating that the boundary layer represents a deficit in mass flow compared to inviscid flow with slip at the wall. It is the distance by which the wall would have to be displaced in the inviscid case to give the same total mass flow as the viscous case. The no-slip condition requires the flow velocity at the surface of a solid object be zero and the fluid temperature be equal to the temperature of the surface. The flow velocity will then increase rapidly within the boundary layer.

The thermal boundary layer thickness is similarly the distance from the body at which the temperature is 99% of the temperature found from an inviscid solution. The ratio of the two thicknesses is governed by the Prandtl number. If the Prandtl number is 1, the two boundary layers are the same thickness. If the Prandtl number is greater than 1, the thermal boundary layer is thinner than the velocity boundary layer. If the Prandtl number is less than 1, which is the case for air at standard conditions, the thermal boundary layer is thicker than the velocity boundary layer.

The General Energy Transport Equation:

$$\underbrace{\frac{\partial(\rho E)}{\partial t}}_{\text{Unsteady}} + \underbrace{\nabla\cdot[\mathbf{V}(\rho E + P)]}_{\text{Convection}} = \nabla\cdot\Big[\underbrace{k_{\text{eff}}\nabla T}_{\text{Conduction}} - \underbrace{\sum_j h_j J_j}_{\substack{\text{Species}\\\text{diffusion}}} + \underbrace{(\bar{\bar{\boldsymbol{\tau}}}_{\text{eff}}\cdot\mathbf{V})}_{\substack{\text{Viscous}\\\text{heating}}}\Big] + \underbrace{S_h}_{\substack{\text{Enthalpy}\\\text{source}}}$$

The above equation represents the general energy transport equation. Energy sources resulting from reactions (endothermic/exothermic) are included for reacting flows. For multiple species flow, energy sources resulting from diffusion are also included in the equation. Energy sources as a function of viscous heating are included which are as follows:

- Thermal energy created by viscous shear in the flow
- Based on Brinkman's number criterion

$$\mathrm{Br} = \frac{\mu U^2}{k\Delta T} \geq 1$$

- These energy sources are not included in the pressure based solver by default. But are included in the density based solver.

Thermal Boundary Conditions:

The boundary conditions can be classified as follows:

1. Neumann Condition (Specified Flux)
2. Robin/Fourier (Specified HTC)
3. Dirichlet (Specified Temperature)

Problem Formulation:

The CFD analysis was done using ANSYS Fluent Release 15.0.

Solver Parameters:

The solver parameters that affect the solution behavior are discussed in this section:

Double Precision Solver:

The Double precision solver is typically designed to reduce the truncation errors in the solution and thus improve the overall heat balance in the solution. As a rule of thumb, the double precision solver should only be enabled in the following situations:

1. Cases of large heat fluxes (Order of MW)

   As in, when there are large, possibly solution dependent heat sources in the energy equation or when there is a widely varying fluid property (functions of temperature) such as nonlinear solids or compressible liquids/gases.

2. Cases when there are large differences in thermal conductivity among materials.

   As in, when the energy numeric becomes stiff or when the flux matching conditions becomes more difficult to be maintained at the solid interface.

Convergence Difficulties:

Every mesh that is setup with specific conditions and solution parameters is expected to converge after a certain number of iterations to ensure the accuracy of the solution. But solution convergence may undergo various difficulties. Many of these difficulties can be recognized based on the following symptoms:

1. Overall imbalance of the heat fluxes.
2. Slow convergence rate (Several Thousand Iterations).
3. Residuals that diverge.
4. Local (cell) temperatures reaching nonphysical values.
5. Skewed cells and improper boundary conditions.

These problems can be avoided by having simple modifications to the solution setup.

Explicit Under-Relaxation:

Advantages:

- Improved Convergence for poor mesh qualities.
- Improved convergence when material properties are strongly dependent on temperature.

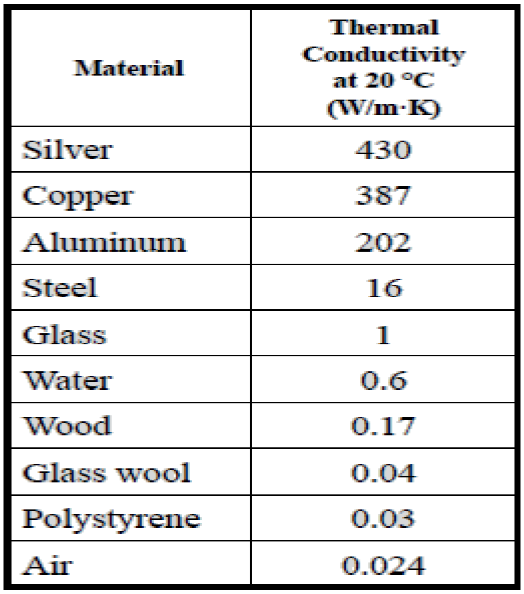
Motivation:

- Energy under-relaxation factor of 1 is often recommended.
- Temperature under relaxation may also be preferred.

Temperature under relaxation factor is generally between 0.25-0.5 and the energy under relaxation factor is generally 1.

Thermal Conductivity of Selected Materials:

Table 2: Thermal Conductivity of materials



Material	Thermal Conductivity at 20 °C (W/m·K)
Silver	430
Copper	387
Aluminum	202
Steel	16
Glass	1
Water	0.6
Wood	0.17
Glass wool	0.04
Polystyrene	0.03
Air	0.024

The tube of the cold plate is given the material properties of copper whereas the cold plate body is given the properties of aluminum.

Conductive Flux Calculation:

This flux is calculated when a situation of having a diffusive flux on the interior of a face exists.

Assumptions:

$\emptyset$ = Temperature

k = Thermal Conductivity

$$D_f = k_f \nabla \emptyset * A$$

$$\sim k_f \, \frac{\emptyset_1 - \emptyset_0}{ds} * \frac{A*A}{A*e_s} + k_f \left\{ \nabla \emptyset * A - \nabla \emptyset * e_s \frac{A*A}{A*e_s} \right\}$$

The flux at the boundary face has a similar expression $\emptyset_1$ is replaced by $\emptyset_f$ and *ds* is replaced by *dr*.

Initial Boundary Conditions:

- Velocity inlet: 0.5 m/s
- Pressure Outlet:
  - Backflow Fluid Temperature = 21 ℃
  - Gauge Pressure = 0 Pa
- Heat Flux:
  - Heat flux at the module surface = 83333.33 W/m^2
  - Total Dissipation targeted = 150 W

Solution Monitors:

Monitors are set to check the progress of the solution through the calculations. In this project 4 solution monitors were setup.

1. Area-weighted static temperature for inlet

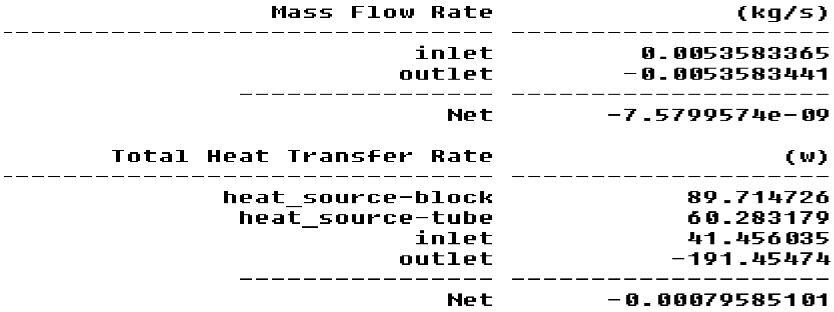
2. Area-weighted static temperature for outlet
3. Area-weighted static pressure for inlet
4. Area-weighted static pressure for outlet

Residuals are set to normalize after every 5 iterations. Normalizing the residuals helps in convergence of the solution.

Solution:

The solution is solved for 1500 iterations and the monitors are updated at an interval of every single iteration.

Energy and Mass Balance:



```
                  Mass Flow Rate                 (kg/s)
-------------------------------- --------------------
                           inlet         0.0053583365
                          outlet        -0.0053583441
                ---------------- --------------------
                             Net       -7.5799574e-09

        Total Heat Transfer Rate                  (w)
-------------------------------- --------------------
               heat_source-block            89.714726
                heat_source-tube            60.283179
                           inlet            41.456035
                          outlet           -191.45474
                ---------------- --------------------
                             Net       -0.00079585101
```

Figure 12: Mass and energy balance

User-Defined Function (UDF):

Fluent is a general CFD analysis software. It is programmed for a general problem solving methodology. So there is a need to be able to customize the boundary conditions. Hence User-defined functions are used.

A UDF is a set of functions written by the user in C language which when interpreted by Fluent can perform all these following functions:

1. Initialization
2. Custom Boundary Condition (both space and time dependent)
3. Material properties
4. Source terms in to any flow equations
5. Chemical reaction rates
6. Post processing of data
7. Add extra equations
8. Mesh motion
9. Discrete phase modification

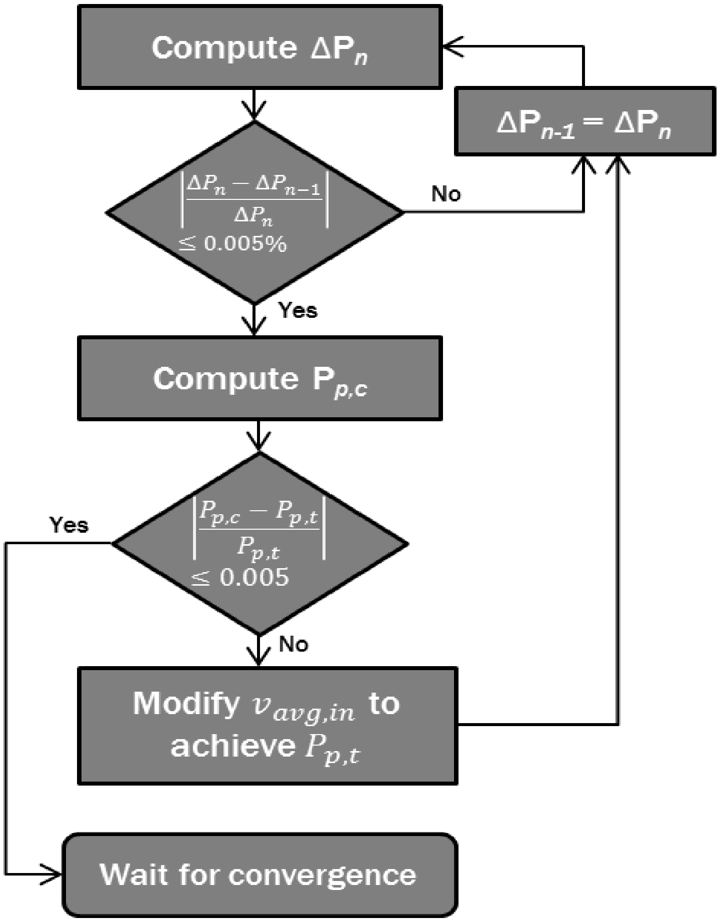


Figure 13: Flowchart for UDF [8]

An interpreted UDF code, shown in the form of a flowchart in figure, is generated which, when coupled to the inlet boundary condition within Fluent®, varies the magnitude of velocity (v_{in}) until the target pumping power (P_p) is reasonably achieved.

In order to better illustrate the inner working of this procedure, let us consider a sample run. The UDF is setup to achieve a P_p value of 21.7W for the base case cold plate configuration. Through use of in-built macros, the solution is setup such that, at each iteration, the pressure drop is compared to its corresponding value from the previous iteration. Once the pressure drop is found to stabilize within a certain predetermined range, referred to as a trigger point (TP), the pumping power P_p is calculated for the current iteration and recorded with v_{in} as TP1. This quantity is compared to the target value and if it is not found to exist within a certain range, the magnitude of average inlet velocity is modified for the first time using the following equation,

$$v_{in,new} = v_{in,old} * f\left(P_{p,c} * P_{p,t}\right)$$

At TP2, the values of P_p and v_{in} are again recorded. However, we do not use this above equation and TP2 to update v_{in} as the two available data points are adequate to setup the following quadratic relation,

$$P_p = Bv_{in}^2 + Cv_{in}$$

There is no constant term in the above relation as we assume that there is zero pumping power for no flow through the cold plate. Using TP1 and TP2, equation is solved to obtain coefficients B and C. The resultant equation is plotted in figure as a dotted-and-dashed red line. At each subsequent trigger point, equation is evaluated using data from the two latest trigger points and solved to update the average inlet velocity. In the current example, the prediction from TP2 and TP3 (dotted green line in figure) is found to be accurate enough that the pumping power at TP4 is within the predefined accuracy (0.5%) and no more changes to

v_{in} are made. The solution is allowed to run to convergence and the results are analyzed. [8]

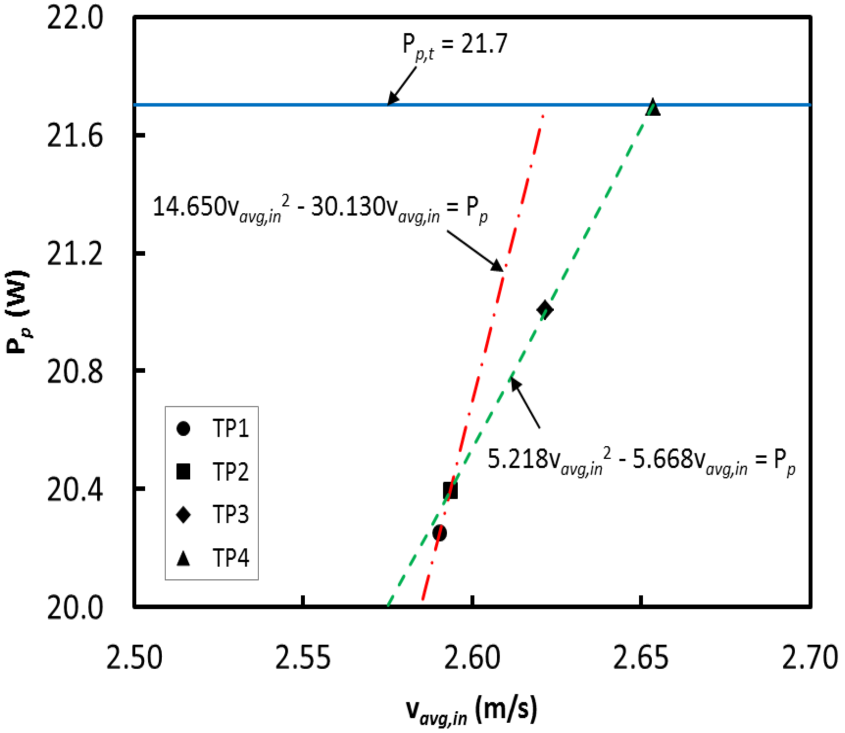


Figure 14: Example depicting UDF principle [8]

Chapter 5

Design of Experiments

Initial Simulation:

The first step is to create the simulation model. The model can be anything from a simple physics problem to a complex multiple conditions and physics coupling. Also in this step we define the parameters to be investigated. The input variables (design parameters) are identified that may include CAD parameters, loading conditions or material properties. The output parameters are chosen from simulation results and may include output pressure, temperature or thermal resistance or can also be custom defined.

Design of Experiments (DOE):

This is a technique used to scientifically determine the location of sampling points and is included as part of Response surface, Goal Driven Optimization, and Six Sigma systems. There are a wide range of Design of Experiments algorithms available today. These techniques all have one common characteristic: they try to locate the sampling points such that the space of random input parameters is explored in the most efficient way, or obtain the required information with a minimum of sampling points. Sample points in efficient locations will not only reduce the required number of sampling points, but also increase the accuracy of the response surface that is derived from the results of the sampling points. By default, the deterministic method uses a central composite design, which combines one center point, points along the axis of the input parameters, and the points determined by a fractional factorial design. Once you have set up your input parameters, you can update the DOE, which submits the generated design points to the analysis system for solution. Design points are solved simultaneously if the analysis system is set up to do so. After the solution is complete, you can update the Response Surface cell, which generates response surfaces for each output parameter based on the data in the generated design points. If you change the Design of Experiments type after doing an initial analysis and preview the Design of Experiments Table, any design points generated for the new algorithm that are the same as design points solved for a previous algorithm will appear as up-to-date. Only the design points that are different from any previously submitted design points need to be solved.

Types of DOE:

Central Composite Design (CCD):

It provides a screening set to determine the overall trends of the meta-model. For each CCD type, the alpha value is defined as the location of the sampling point that accounts for all quadratic main effects. Following properties are associated with CCD DOE:

  - Face-centered: Alpha value equals 1.0. It is a three level design with no rotatability.
  - Rotatable: Alpha value is calculated based on the input variables and fraction of the fractional part. It is a five level design that includes rotatability. It has the same variance as of the fitted value regardless of the direction from the center point.
  - VIF-Optimality: A five-level design in which the alpha value is calculated by minimizing a measure of non-orthogonality known as the Variance Inflation Factor (VIF). The more highly correlated the input variable with one or more terms in a regression model, the higher the Variance Inflation Factor.
  - G-Optimality: Minimizes the largest expected variance of prediction over the region of interest.
  - Auto-Defined

Optimal Space-Filling Design (OSF):

Optimal Space-Filling Design (OSF) creates optimal space filling Design of Experiments (DOE) plans according to some specified criteria. Essentially, OSF is a Latin Hypercube Sampling Design (LHS) that is extended with post-processing. It is initialized as an LHS and then optimized several times, remaining a valid LHS (without points sharing rows or columns) while achieving a more uniform space distribution of points (maximizing the distance between points). Optimal Space-Filling (OSF) design is able to distribute the design parameters equally throughout the design space with the objective of gaining the maximum insight into the design with the fewest number of points. This advantage makes it appropriate when a more complex meta-modeling technique such as Kriging, Non-Parametric Regression or Neural Networks is used. OSF shares some of the same disadvantages as LHS, though to a lesser degree. Possible disadvantages of an OSF design are that extremes (i.e., the corners of the design space) are not necessarily covered and that the selection of too few design points can result in a lower quality of response prediction. The properties associated with OSF are as follows:

- Design Type:
  - Max-Min Distance
  - Centered L2
  - Maximum Entropy
- Maximum Number of cycles: Determines the number of optimization loops the algorithm needs, which in turns determines the discrepancy of the DOE.
- Sample Types:
  - CCD Samples
  - Linear Model Samples

    - Pure Quadratic Model Samples
    - Full Quadratic Samples
    - User-Defined Samples
- Seed Value: This property allows you to generate different samplings (by changing the value) or to regenerate the same sampling (by keeping the same value). Defaults to 0.
- Number of Samples

Box-Behnken Design:

A Box-Behnken Design is a three-level quadratic design that does not contain any fractional factorial design. The sample combinations are treated in such a way that they are located at midpoints of edges formed by any two factors. The design is rotatable. One advantage of a Box-Behnken design is that it requires fewer design points than a full factorial CCD. Additionally, a Box-Behnken Design avoids extremes, allowing you to work around extreme factor combinations. Consider using the Box-Behnken Design DOE type if your project has parametric extremes. Since the Box-Behnken DOE doesn’t have corners and does not combine parametric extremes, it can reduce the risk of update failures. Possible disadvantages of a Box-Behnken design are that prediction at the corners of the design space is poor and that there are only three levels per parameter. No additional properties are available for the Box-Behnken Design DOE type.

Custom:

The Custom DOE type allows for definition of a custom DOE Table. You can manually add new design points, entering the input and (optionally) output parameter values directly into the table. If you previously solved the DOE using one of the other algorithms, those design points will be retained and you can add new design points to the table. You can also import and export design points into the custom DOE Table from the Parameter Set.

Custom + Sampling:

The Custom + Sampling DOE type provides the same capabilities as the Custom DOE type and allows you to complete the DOE table automatically to fill the design space efficiently. The generation of these new design points takes into account the coordinates of previous design points. The property associated with this type is:

- Total Number of Samples: If the total number of samples is less than the number of existing points, no any new points will be added. If there are discrete input parameters, the total number of samples corresponds to the number of points that should be reached for each combination of discrete parameters.

Sparse Grid Initialization:

Sparse Grid Initialization is the DOE type required to run a Sparse Grid Interpolation. Sparse Grid is an adaptive meta-model driven by the users accuracy request. It increases the accuracy of the response surface by automatically refining the matrix of design points in locations where the relative error of the output parameter is higher. This DOE type generates the levels 0 and 1 of the Clenshaw-Curtis Grid. In other words, because the Sparse Grid algorithm is based on a hierarchy of grids, the Sparse Grid Initialization DOE type generates a DOE matrix containing all the design points for the smallest required grid: the level 0 (the point at the current values) plus the level 1 (two points per input parameters). One advantage to a Sparse Grid design is that it refines only in the directions necessary, so that fewer design points are needed for the same quality response surface. Another is that Sparse Grid is effective at handling discontinuities. Although this DOE type is required to build a Sparse Grid response surface, it can also be used by other types of response surface. No additional properties are available for the Sparse Grid Initialization DOE type.

Latin Hypercube Sampling Design (LHS):

The DOE is generated by the LHS algorithm, an advanced form of the Monte Carlo sampling method that avoids clustering samples. In a Latin Hypercube Sampling, the points are randomly generated in a square grid across the design space, but no two points share the same value. Possible disadvantages of an LHS design are that extremes are not necessarily covered and that the selection of too few design points can result in a lower quality of response prediction. Properties associated with this type of DOE are as follows:

- Sample Types:
    - CCD samples
    - Linear model samples
    - Pure quadratic model samples
    - Full quadratic samples
    - User defined samples
- Seed Value: Set the value used to initialize the random number generator invoked internally by the LHS algorithm.
- Number of Samples

In this cold plate analysis, amongst the types explained above, Optimal Space-Filling Design (OSF) type of DOE was used. Among the properties associated, maximum entropy was selected. Number of cycles was 20, sample type was full quadratic model samples and the seed value was given a value 0.

Chapter 6

Results and Analysis

Temperature Contours:

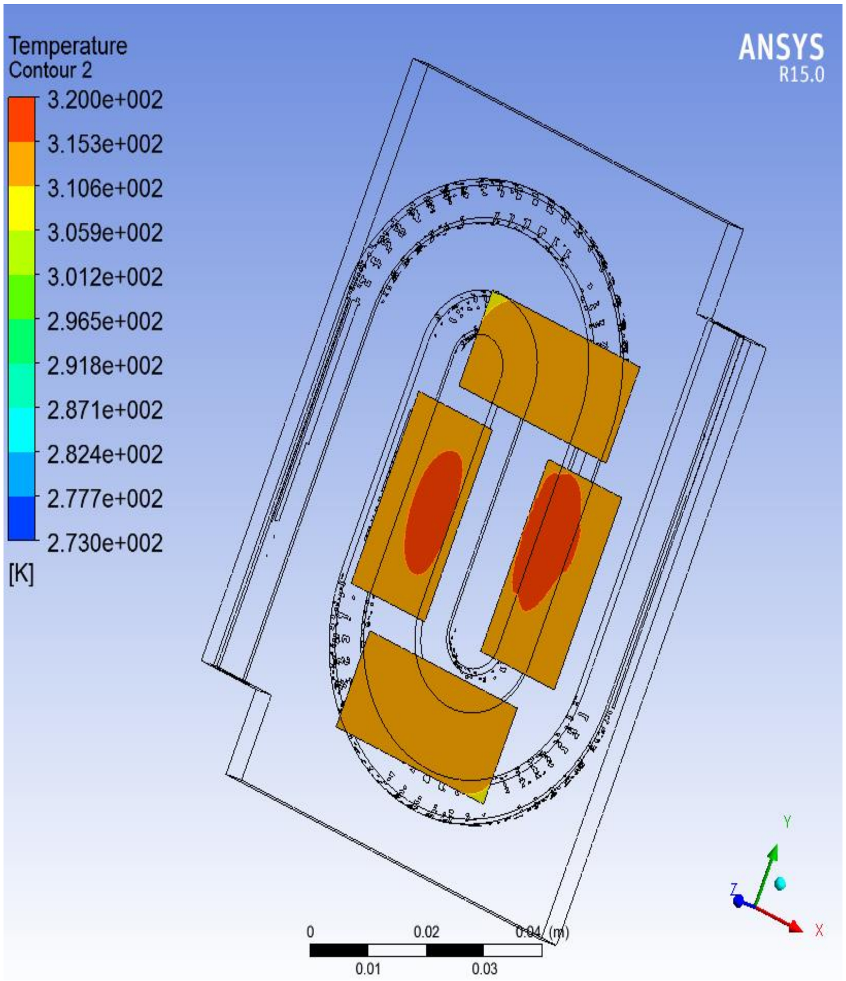


Figure 15: Contour for temperature at the module surface

This contour shows the temperature variation at the interface between the cold plate and the module surface. The temperature range is 273 K – 320 K.

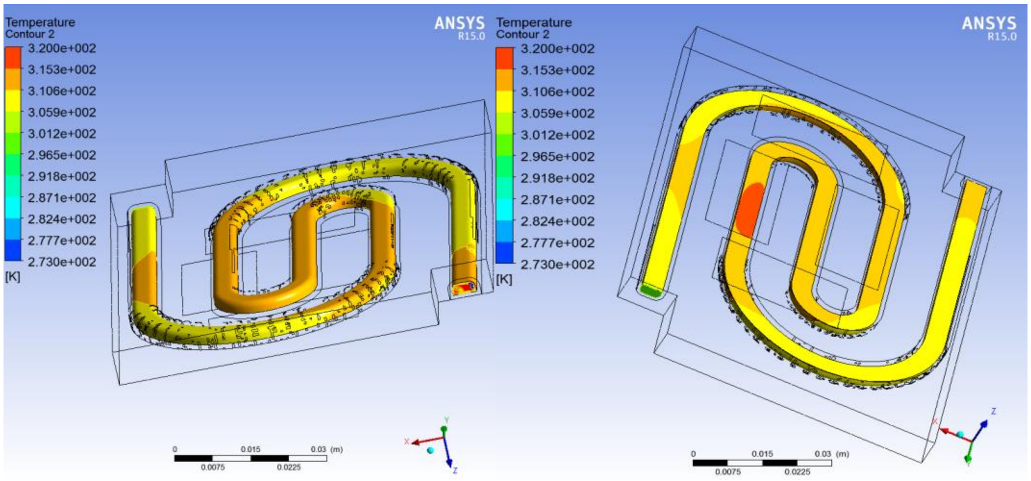


Figure 16: Temperature contour at the fluid region

These temperature plots are for the fluid region in the tube and the inlet and the outlet as well. There is hotspot formulation at the module and the cold plate interface as the heat flux applied is assumed to only in the direction normal to the surface and the other components of the heat flux are considered to be negligible. The temperature range is the same 273 K – 320 K.

Since just the server level cooling level is concerned, the max temperature is well under the acceptable limits.

Response Surface:

The Response Surfaces are functions of different nature where the output parameters are described in terms of the input parameters. They are built from the

Design of Experiments in order to provide quickly the approximated values of the output parameters, everywhere in the analyzed design space, without a need to perform a complete solution. The accuracy of a response surface depends on several factors: complexity of the variations of the solution, number of points in the original Design of Experiments and choice of the response surface type. ANSYS Design Xplorer provides tools to estimate and improve the quality of the response surfaces. Once response surfaces are built, you can create and manage response points and charts. These post processing tools allow exploring the design and understanding how each output parameter is driven by input parameters and how the design can be modified to improve its performances.

The following are the types of meta-models that can be deployed to create the response surface based on the DOE design points.

- Standard Response Surface – Full 2nd order polynomial
- Kriging
- Non Parametric regression
- Neural Network
- Sparse Grid

Kriging:

Kriging is a meta-modeling algorithm that provides an improved response quality and fits higher order variations of the output parameter. It is an accurate multidimensional interpolation combining a polynomial model similar to the one of the standard response surface, which provides a “global” model of the design space, plus local deviations determined so that the Kriging model interpolates the DOE points.

The response surface is updated with the kriging algorithm. Verification points are generated to verify that the predicted v/s observed curve fits in the goodness of fit line.

As we can see in the figure; If the accuracy of the verification points is larger than the Predicted Relative Error given by Kriging, you can insert the verification points as refinement points (this must be done in manual refinement mode) and then run a new auto-refinement so that the new points will be included in the generation of the Response Surface.

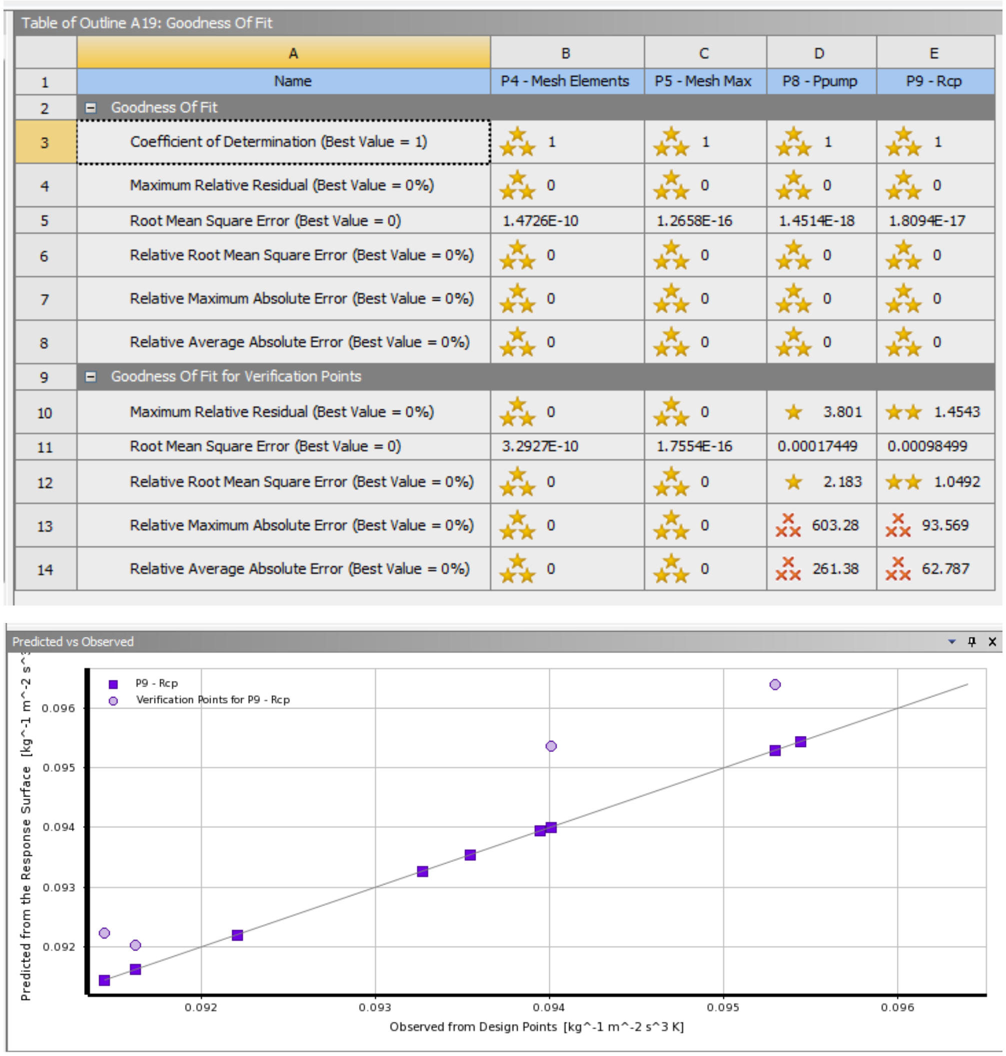


Table of Outline A19: Goodness Of Fit

	A	B	C	D	E
1	Name	P4 - Mesh Elements	P5 - Mesh Max	P8 - Ppump	P9 - Rcp
2	⊟ Goodness Of Fit				
3	Coefficient of Determination (Best Value = 1)	★★★ 1	★★★ 1	★★★ 1	★★★ 1
4	Maximum Relative Residual (Best Value = 0%)	★★★ 0	★★★ 0	★★★ 0	★★★ 0
5	Root Mean Square Error (Best Value = 0)	1.4726E-10	1.2658E-16	1.4514E-18	1.8094E-17
6	Relative Root Mean Square Error (Best Value = 0%)	★★★ 0	★★★ 0	★★★ 0	★★★ 0
7	Relative Maximum Absolute Error (Best Value = 0%)	★★★ 0	★★★ 0	★★★ 0	★★★ 0
8	Relative Average Absolute Error (Best Value = 0%)	★★★ 0	★★★ 0	★★★ 0	★★★ 0
9	⊟ Goodness Of Fit for Verification Points				
10	Maximum Relative Residual (Best Value = 0%)	★★★ 0	★★★ 0	★ 3.801	★★ 1.4543
11	Root Mean Square Error (Best Value = 0)	3.2927E-10	1.7554E-16	0.00017449	0.00098499
12	Relative Root Mean Square Error (Best Value = 0%)	★★★ 0	★★★ 0	★ 2.183	★★ 1.0492
13	Relative Maximum Absolute Error (Best Value = 0%)	★★★ 0	★★★ 0	✗✗✗ 603.28	✗✗✗ 93.569
14	Relative Average Absolute Error (Best Value = 0%)	★★★ 0	★★★ 0	✗✗✗ 261.38	✗✗✗ 62.787

Figure 17: (a)GOD for response surface (b) predicted v/s observed curve

Refinement:

The Refinement properties in the Response Surface Properties view determine the number and the spread of the refinement points. The properties are as follows:

- Maximum number of refinement points: Indicates the max number of verification points that can be generated using kriging algorithm
- Number of refinement points: Indicates the existing number of refinement points
- Maximum predicted relative error: Indicates the relative error acceptable to all parameters
- Predicted relative error: Indicates the predicted relative error
- Convergence: Yes/No

Since the points generated by the kriging algorithm do not fit the goodness of fit line, refinement is done for those points.

Advanced Refinement Options:

- Maximum output: Only the output with the largest Predicted Relative Error is considered. Only one refinement point is generated in each iteration.
- All outputs: All outputs are considered. Multiple refinement points are generated in each iteration.
- Sum of outputs: The combined Predicted Relative Error of all outputs is considered. Only one refinement point is generated in each iteration.

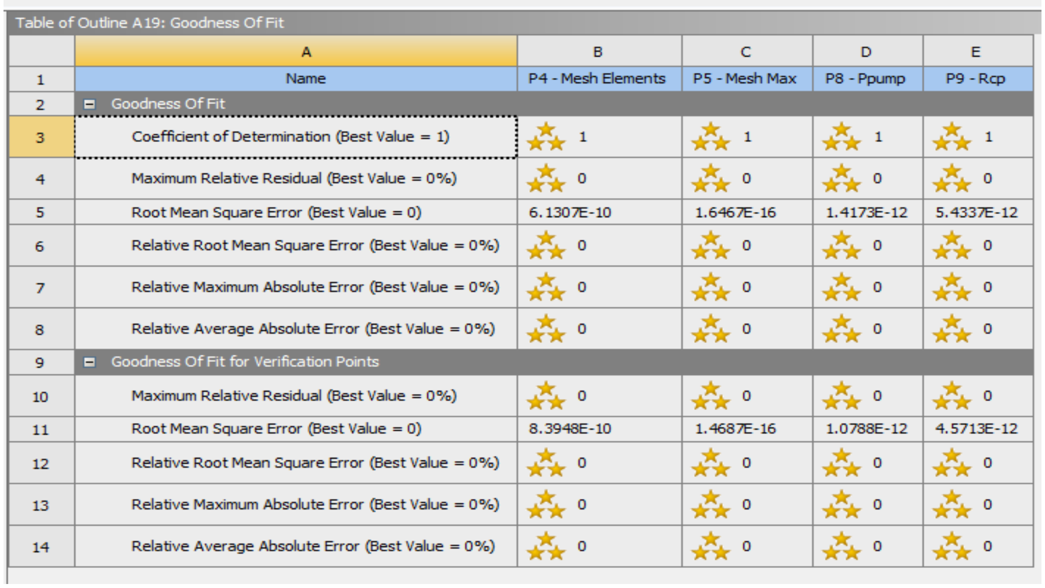


Table of Outline A19: Goodness Of Fit

	A	B	C	D	E
1	Name	P4 - Mesh Elements	P5 - Mesh Max	P8 - Ppump	P9 - Rcp
2	⊟ Goodness Of Fit				
3	Coefficient of Determination (Best Value = 1)	★★★ 1	★★★ 1	★★★ 1	★★★ 1
4	Maximum Relative Residual (Best Value = 0%)	★★★ 0	★★★ 0	★★★ 0	★★★ 0
5	Root Mean Square Error (Best Value = 0)	6.1307E-10	1.6467E-16	1.4173E-12	5.4337E-12
6	Relative Root Mean Square Error (Best Value = 0%)	★★★ 0	★★★ 0	★★★ 0	★★★ 0
7	Relative Maximum Absolute Error (Best Value = 0%)	★★★ 0	★★★ 0	★★★ 0	★★★ 0
8	Relative Average Absolute Error (Best Value = 0%)	★★★ 0	★★★ 0	★★★ 0	★★★ 0
9	⊟ Goodness Of Fit for Verification Points				
10	Maximum Relative Residual (Best Value = 0%)	★★★ 0	★★★ 0	★★★ 0	★★★ 0
11	Root Mean Square Error (Best Value = 0)	8.3948E-10	1.4687E-16	1.0788E-12	4.5713E-12
12	Relative Root Mean Square Error (Best Value = 0%)	★★★ 0	★★★ 0	★★★ 0	★★★ 0
13	Relative Maximum Absolute Error (Best Value = 0%)	★★★ 0	★★★ 0	★★★ 0	★★★ 0
14	Relative Average Absolute Error (Best Value = 0%)	★★★ 0	★★★ 0	★★★ 0	★★★ 0

Figure 18: GOD for refinement of verification points

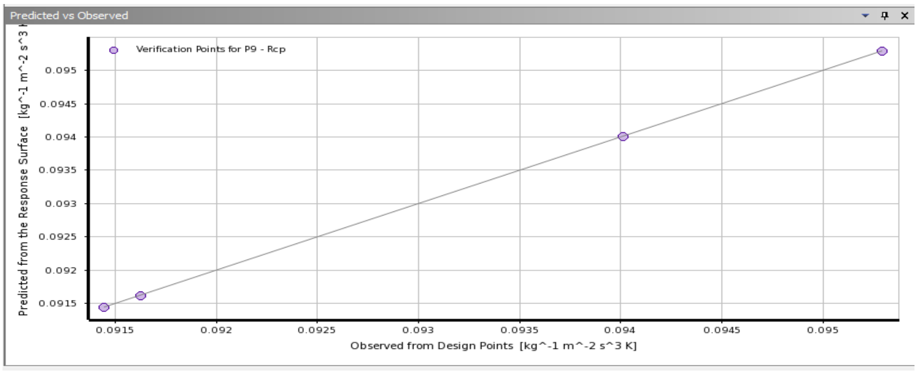


Figure 19: Predicted v/s observed for the refinement module

Goodness of Fit and Goodness of Fit Criterion:

Goodness of fit is often used as a sanity check for the Meta model deployed for the generation of the response surface. If the goodness of fit is not of the expected quality, the Meta model can me modified and refinement be done to get the verification points to be of the expected quality.

Criterion:

- Co-efficient of determination (R^2) :

  The percent of the variation of the output parameter that can be explained by the response surface regression equation. That is, the Coefficient of Determination is the ratio of the explained variation to the total variation. The best value is 1.

  Mathematically expressed as,

$$1 - \frac{\sum_{i=1}^{N}(y_i - \dot{y}_i)^2}{\sum_{i=1}^{N}(y_i - \bar{y}_i)^2}$$

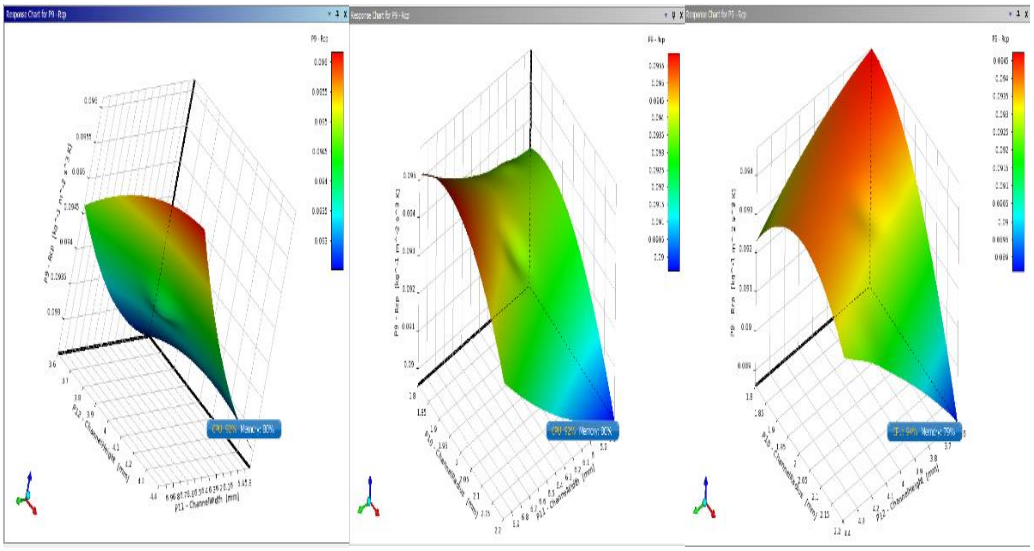


Figure 20: Response Surface with respect to input parameters

Local Sensitivity:

Local Sensitivity charts allow you to see the impact of continuous input parameters (both with and without manufacturable Values) on output parameters. At the Response Surface level, sensitivity charts are “Single Parameter Sensitivities.” This means that design exploration calculates the change of the output(s) based on the change of inputs independently, at the current value of each input parameter. The larger the change of the output parameter(s), the more significant is the role of the input parameters that were varied. As such, single parameter sensitivities are local sensitivities.

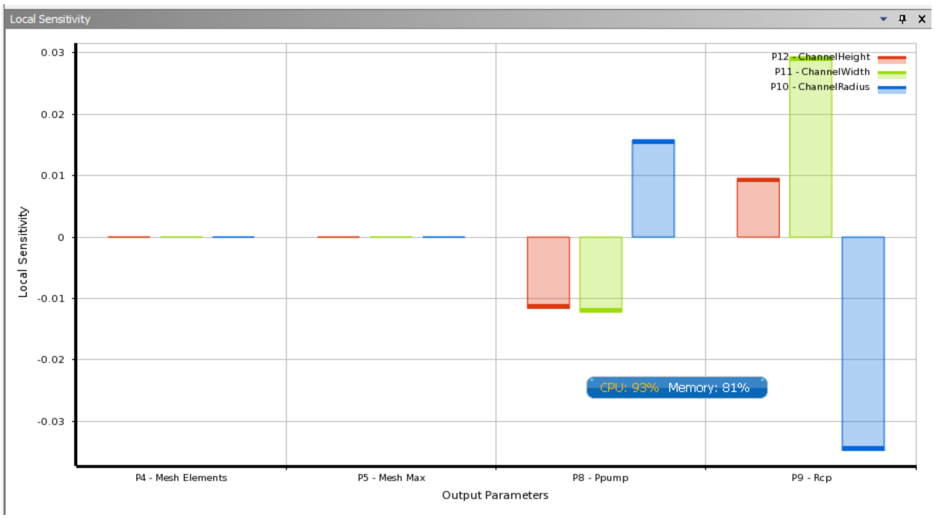


Figure 21: Local Sensitivity chart

Response Surface Optimization:

After the response surface is obtained from the new refinement verified points, the project can be setup for a design optimization. In this case a goal driven optimization.

Optimization Methods:

- Screening
- Nonlinear programming by quadratic lagrangian
- Mixed-integer sequential quadratic programming
- Multi-objective genetic algorithm
- Adaptive single objective
- Adaptive multi objective

The method used for this optimization study was screening method which is the default method.

Screening method:

It allows you to generate a new sample set and sort its samples based on objectives and constraints. It is a non-iterative approach that is available for all types of input parameters. Usually the Screening approach is used for preliminary design, which may lead you to apply the MOGA or NLPQL options for more refined optimization results.

Constraint: Minimize thermal resistance

Generate Candidate points: 4 candidates

Table 3: Candidate Points from the optimization

CANDIDATE POINTS	**OBSERVED R_{cp} VALUES**
Point 1	0.094013
Point 2	0.091627
Point 3	0.095296
Point 4	0.091447

Chapter 7

Conclusion

Multi variable optimization for the 3 design variables; channel width, channel height and the channel radius; was successfully carried out in this project. The candidate point 4 has an 11.661% reduction in thermal resistance over the base line case. This is significant as an 11.6% reduction can be ultimately translated in extendibility of the cooling technology without impacting the power. Hence, we can have the same design for the cold plate but for a higher power dissipation. More complex computing applications can be thermally managed using this cold plate design.

References

1. Ellsworth, Jr., M.J., Campbell, L.A., Simons, R.E., Iyengar, M.K., Schmidt, R.R., Chu, R.C., “The Evolution of Water Cooling for IBM Large Server Systems: Back to the Future,” proceedings of the 2008 Intersociety Conference on Thermal and Thermomechanical Phenomena in Electronics Systems (ITherm) Conference, Orlando, FL, USA, May 28-31.
2. Simons. R.E.,”The Evolution of IBM High Performance Cooling Technology,” Proceedings of the Eleventh Annual IEEE Semiconductor Thermal Measurement and Management Symposium, 1995, pp. 102-112.
3. Ellsworth Jr. M.J.; Iyengar, M.K., “Energy Efficiency Analyses and Comparison of Air and Water Cooled High Performance Servers,” proceedings of the 2009 InterPACK Conference, San Francisco, CA., July 19-23.
4. https://en.wikipedia.org/wiki/File:Heat_sink_control_volume.png
5. https://en.wikipedia.org/wiki/File:Difference_between_thermal_conductivity_of_thermal_interface_materials_and_thermal_contact_resistance.png
6. Gary F. Goth, Amilcar Arvelo, Jason Eagle, Michael J. Ellsworth, Jr.1, Kenneth C. Marston,Arvind K. Sinha, Jeffrey A. Zitz, “Thermal and Mechanical Analysis and Design of the IBM Power 775 Water Cooled Supercomputing Central Electronics Complex,” IBM Corporation.

7. Michael J. Ellsworth Jr., Levi Campbell, "Technical Brief: Design Considerations for High Performance Processor Liquid Cooled Cold Plates," electronics cooling, issue December 22nd 2011.
8. John Fernandes, Saeed Ghalambor, Dereje Agonafer, Vinod Kamath, Roger Schmidt, "Multi-Design Variable Optimization for a Fixed Pumping Power of a Water-Cooled Cold Plate for High Power Electronics Applications," ITherm 2012.

Biographical Information

Pratik Kekre is a graduate student at the University of Texas at Arlington. He defended his Masters in Spring 2014. He has completed his Bachelor's of engineering from Pune University, India in mechanical engineering.

Pratik has a special interest towards thermal management of high end electronic devices. He has worked with professor Dr. Dereje Agonafer at EMNSPC (Electronics Mems & Nanoelectronics systems packaging center) at UTA on various projects focusing on the server level of data center cooling. His interests are basically fluid dynamics, computational fluid dynamics, thermodynamics and heat transfer.