# 2017

Unison controls limited

Mohak Amishbhai shah

Summer internship report

# DEVELOPMENT OF FINITE ELEMENT MODEL FOR THE STUDY HEAT TRANSFER IN HEATSINK AND CREATING TRANSIENT THERMAL ANALYSIS OF HEATSINKS USING ANSYS

**Institution : L.D. COLLEGE OF ENGINNERING**
**YEAR : 3rd**
**Discipline : Mechanical**

# Synopsis

As performance limits have been pushed further by need to have compact and integrated circuits, heat dissipation becomes critical parameter. Tightly packed and integrated circuit means less space for heat to escape which leads to heat accumulation and thermal stress in product. Conventionally heat dissipation is controlled by the virtue of heatsink, so while designing competent heatsink for such circuits we need consider more cautious approach. thermal management have been part of analysis for many years , but with growing need of accuracy many companies have opted for software approach ,in addition to that they have also started relying on CFD and FEM analysis, which have evolved over the past few years. These analysis not just bring accuracy, speed but fusion of fundamental principle and practical boundary condition. Another most important factor which this software has to offer is quantification of result, which is processed further to find cause of result, so in a manner this project make us analyze our product's capacity Here in this project Ansys thermal software is used for analysis.

## Objective:

Following objectives are fulfilled during this project

- Acquire the fundamentals aspect regarding heat transfer in heatsinks
- Obtain the basics of thermal simulation in ansys software
- Develop 3D thermal profile of various heatsinks using ansys
- Compare ansys transient results of heatsink with practical result

This project is useful for any device running on high power, facing challenges regarding thermal management as well as regulation of temperature. So in way it is useful in predicting cooling pattern, thermal stress in a component.

# Table of Contents

## LIST OF FIGURES

# CHAPTER 1

# Introduction to Heatsinks

The primary purpose of heat sink is to reduce the temperature of the electronic component attached to it where excess heat is transferred into another medium, typically air thus heat-related failures of electronic components are prevented. There are several design aspects which are important in order to ensure the efficient heat dissipation through heat sink. Some of them are related to operating conditions, such as artificial air circulation, choice of thermal interface materials, or a proper attachment to the heat-emitting component. However, most of the factors which determine the efficiency of heat sink are related to its construction, e.g. larger active surface area, type of material it is manufactured

Heatsinks classified on various basis such as geometry, nature of cooling, types of source etc. Classification of our concern is on the basis of cooling which has been given below:

1) Natural cooled Heatsink :

   Heatsinks are exposed to surrounding air at ambient temperature.

2) Forced cooled heatsink:

   Heat transfer takes place by the fluid in motion by external sources such as pump, fan etc.

In heatsinks transfer of heat is primarily occurs through conduction and convection, where rate primarily depends on temperature difference, material of heatsinks, nature of cooling

To understand the principle of a heat sink, consider Fourier's law of heat conduction. Fourier's law of heat conduction, simplified to a one-dimensional form in the x-direction, shows 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 is proportional to the product of the temperature gradient and the cross-sectional area through which heat is transferred.

$$Q = -kA\frac{\Delta T}{\Delta x} \qquad (1.1)$$

Where

Q= heat transferred

K = thermal conductivity

A = cross-section area

$\Delta T/\Delta x$ = temperature gradient

Sketch of a heat sink in a duct used to calculate the governing equations from conservation of energy and Newton's law of cooling.

Consider a heat sink in a duct, where air flows through the duct. It is assumed that the heat sink base is higher in temperature than the air. Applying the conservation of energy, for steady-state conditions, and Newton's law of cooling to the temperature nodes shown in the diagram gives the following set of equations:

$$Q = mc_p(T_{air,in} - T_{air.out}) \qquad (1.2)$$

$$Q = \frac{T_{hs} - T_{av}}{R_{hs}}$$

Where

$$T_{av} = \frac{T_{air,in} + T_{air,out}}{2}$$

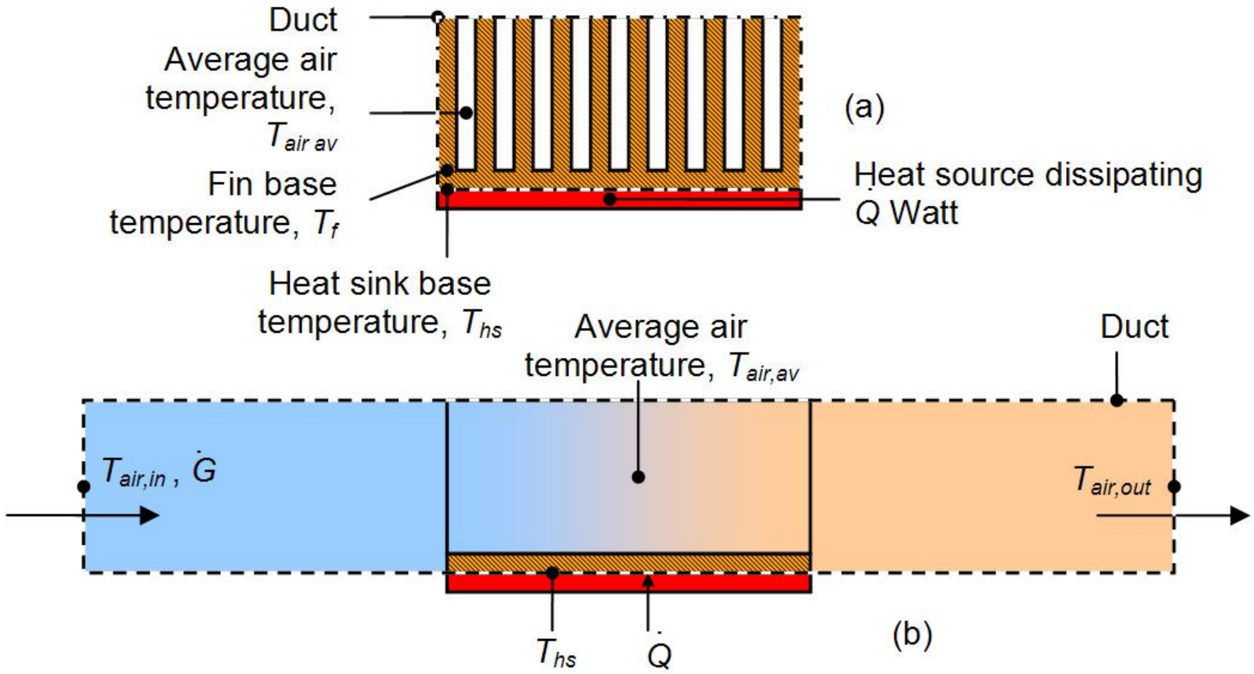



**Figure 1 heat flow in heat sink**

Using the mean air temperature is an assumption that is valid for relatively short heat sinks. When compact heat exchangers are calculated, the logarithmic mean air temperature is used. m is the air mass flow rate in kg/s.

The above equations show that when the air flow through the heat sink decreases, this results in an increase in the average air temperature. This in turn increases the heat sink base temperature, and additionally the thermal resistance of the heat sink will also increase which result is a higher heat sink base temperature.

The inlet air temperature relates strongly with the heat sink base temperature. For example, if there is recirculation of air in a product, the inlet air temperature is not the ambient air temperature. The inlet air temperature of the heat sink is therefore higher, which also results in a higher heat sink base temperature. If there is no air flow around the heat sink, energy cannot be transferred.

Natural convection requires free flow of air over the heat sink. If fins are not aligned vertically, or if fins are too close together to allow sufficient air flow between them, the efficiency of the heat sink will decline.

## 1.1 Design factors: Thermal resistance

For semiconductor devices used in a variety of consumer and industrial electronics, the idea of thermal resistance simplifies the selection of heat sinks. The heat flow between the semiconductor die and ambient air is modeled as a series of resistances to heat flow; there is a resistance from the die to the device case, from the case to the heat sink, and from the heat sink to the ambient air. The sum of these resistances is the total thermal resistance from the die to the ambient air. Thermal resistance is defined as temperature rise per unit of power, analogous to electrical resistance, and is expressed in units of degrees Celsius per watt (°C/W). If the device dissipation in watts is known, and the total thermal resistance is calculated, the temperature rise of the die over the ambient air can be calculated.

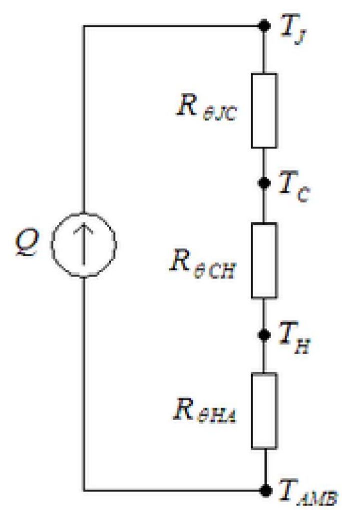



**Figure 2 electrical analogy of thermal resistance**

The idea of thermal resistance of a semiconductor heat sink is an approximation. It does not take into account non-uniform distribution of heat over a device or heat sink. It only models a system in thermal equilibrium, and does not take into account the change in temperatures with time, nor does it reflect the non-linearity of radiation and convection with respect to temperature rise. However, manufacturers tabulate typical values of thermal resistance for heat sinks and semiconductor devices, which allows selection of commercially manufactured heat sinks to be simplified.

Consider a component such as a silicon transistor that is bolted to the metal frame of a piece of equipment. The transistor's manufacturer will specify parameters in the datasheet called the absolute thermal resistance from junction to case ($R_{\theta JC}$ ), and the maximum allowable temperature of the semiconductor junction ($T_{JMAX}$ ). The specification for the design should include a maximum temperature at which the circuit should function correctly. Finally, the designer should consider how the heat from the transistor will escape to the environment: this might be by convection into the air, with or without the aid of a heatsink, or by conduction through the printed circuit board. For simplicity, let us assume that the designer decides to bolt the transistor to a metal surface (or heatsink) that is guaranteed to be less than  above the ambient temperature.

Note: $T_{HS}$ appears to be undefined.

Given all this information, the designer can construct a model of the heat flow from the semiconductor junction, where the heat is generated, to the outside world. In our example, the heat has to flow from the junction to the case of the transistor, then from the case to the metalwork. We do not need to consider where the heat goes after that, because we are told that the metalwork will conduct heat fast enough to keep the temperature less than $\Delta T_{hs}$ above ambient: this is all we need to know.

Suppose the engineer wishes to know how much power he can put into the transistor before it overheats. The calculations are as follows.

Total absolute thermal resistance from junction to ambient = $R_{\theta JC} + R_{\theta B}$

Where $R_{\theta B}$ is the absolute thermal resistance of the bond between the transistor's case and the metalwork. This figure depends on the nature of the bond - for example, a thermal bonding pad or thermal transfer grease might be used to reduce the absolute thermal resistance.

Maximum temperature drop from junction to ambient = $T_{Jmax} - (T_{amb} + \Delta T_{hs})$.

We use the general principle that the temperature drop $\Delta T$ across a given absolute thermal resistance $R_\theta$ with a given heat flow $Q$ through it is:

$$\Delta T = Q \times R_\theta \tag{1.1.1}$$

Substituting our own symbols into this formula gives:

$$T_{Jmax} - (T_{amb} + \Delta T_{hs}) = Q_{max} \left( R_{\theta JC} + R_{\theta B} + R_{\theta HA} \right) \tag{1.1.2}$$

And rearranging,

$$Q_{max} = \frac{T_{Jmax} - (T_{amb} + \Delta T_{hs})}{\left( R_{\theta JC} + R_{\theta B} + R_{\theta HA} \right)} \tag{1.1.3}$$

The designer now knows $Q_{max}$ the maximum power that the transistor can be allowed to dissipate, so he can design the circuit to limit the temperature of the transistor to a safe level.

Let us substitute some sample numbers:

$T_{Jmax} = 125°\text{C}$ (Typical for a silicon transistor)

$T_{amb} = 21°\text{C}$ (a typical specification for commercial equipment)

$R_{\theta JC} = 1.5 \ {}^{°\text{C}}/_{W}$ (For a typical TO-220 package)

$R_{\theta B} = 0.1 \ {}^{°\text{C}}/_{W}$ (a typical value for an elastomeric heat-transfer pad for a TO220 Package)

$R_{\theta HA} = 4 \ {}^{°\text{C}}/_{W}$ (a typical value for a heatsink for a TO-220 package])

The result is then

$$Q_{max} = \frac{125 - 21}{(1.5 + .1 + 4)}$$

$$Q_{max} = 18.6\, W$$

This means that the transistor can dissipate about 18 watts before it overheats. A cautious designer would operate the transistor at a lower power level to increase its reliability

This method can be generalized to include any number of layers of heat-conducting materials, simply by adding together the absolute thermal resistances of the layers and the temperature drops across the layers.

## 1.2 Material of Heatsink

The most common heat sink materials are aluminum alloys. Aluminum alloy 1050A has one of the higher thermal conductivity values at 229 W/m•K [7] but is mechanically soft. Aluminum alloys 6060 (low stress), 6061 and 6063 are commonly used, with thermal conductivity values of 166 and 201 W/m•K, respectively. The values depend on the temper of the alloy.

Copper has excellent heat sink properties in terms of its thermal conductivity, corrosion resistance, biofouling resistance, and antimicrobial resistance (see Main Article: Copper in heat exchangers). Copper has around twice the thermal conductivity of aluminum and faster, more efficient heat absorption. Its main applications are in industrial facilities, power plants, solar thermal water systems, HVAC systems, gas water heaters, forced air heating and cooling systems, geothermal heating and cooling, and electronic systems.

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 heat sinks can be extruded, but the less ductile copper cannot.

Diamond is another heat sink material and its thermal conductivity of 2000 W/m•K 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. For thermal management applications, the outstanding thermal conductivity and diffusivity of diamond is an essential. Nowadays synthetic diamond is used as submounts for high-power integrated circuits and laser diodes.

Composite materials can be used. Examples are a copper-tungsten pseudoalloy, AlSiC (silicon carbide in aluminum matrix), Dymalloy (diamond in copper-silver alloy matrix), and E-Material (beryllium oxide in beryllium matrix). Such materials are often used as substrates for chips, as their thermal expansion coefficient can be matched to ceramics and semiconductors.

## 1.3 Fin efficiency

Fin efficiency is one of the parameters which make a higher thermal conductivity material important. A fin of a heat sink may be considered to be a flat plate with heat flowing in one end and being dissipated into the surrounding fluid as it travels to the other. As heat flows through the fin, the combination of the thermal resistance of the heat sink impeding the flow and the heat lost due to convection, the temperature of the fin and, therefore, the heat transfer to the fluid, will decrease from the base to the end of the fin. Fin efficiency is defined as the actual heat transferred by the fin, divided by the heat transfer were the fin to be isothermal (hypothetically the fin having infinite thermal conductivity). Equations 6 and 7 are applicable for straight fins.

$$n_f = \frac{\tanh(mL_C)}{mL_C} \tag{1.3.1}$$

Where $$mL_C = \sqrt{\frac{2h_f}{kt_f}}L_f$$

$h_f$ Is the convection coefficient of the fin

Air: 10 to 100 $\mathrm{W/m^2K}$

Water: 500 to 10,000 $\mathrm{W/m^2K}$

K is the thermal conductivity of the fin material

Aluminium: 120 to 240 W/(m·K)

$L_f$ is the fin height (m)

$t_f$ is the fin thickness (m)

Fin efficiency is increased by decreasing the fin aspect ratio (making them thicker or shorter), or by using more conductive material (copper instead of aluminium, for example).

# CHAPTER 2

## Introduction to thermal analysis

Failures of any electronic component or device is directly related to its operating temperature Increased temperature shortens normal lifetime. Its exact effect can be determined mathematically; one possible method is by using Arrhenius equation, which relates how increased temperature accelerates the age of a product compared to its normal operating temperature

$$A_f = Ae^{\left(\frac{Ea}{k}\left\{\frac{1}{Tu} - \frac{1}{Tt}\right\}\right)} \tag{2.1}$$

Where

$A_f$ – Acceleration factor,

A – A proportional multiplier, which can be a function of temperature,

i.e. A=A (t),

$E_a$ – Activation energy in electron-volts (eV) is related to physical mechanism of electrical failure [3]

k – Boltzmann's constant (k=8.617·10-5 eV/$T_k$),

$T_k$– temperature K,

$T_u$– Reference temperature K,

$T_t$– test temperature

In practice such dependency means, that lowering the temperature by 10ºC potentially doubles the reliability of the product; and vice versa: increasing temperature by 10ºC decreases the lifetime by 50%. This is a solid motivation for development of better cooling solutions for electronic components.

Conduction, radiation and convection are the three fundamental mechanisms which define the distribution of thermal energy. As stated by first law of thermodynamics, such thermal processes take place when there is a difference in temperature, in our case between particular electronic component and ambient air. These processes follow the principle of conservation of energy and internal energy of modeled object as a logical quantity to solve for. However, in practice it is expressed in terms of temperature, since this parameter is more easy and convenient to measure.

## 2.1 Thermal analysis using software

Any thermal software is embedded with certain governing equation followed by numerical methods to give realistic solution. Equation which governs thermal solution is listed below:

Coordinates: (x,y,z)  Time : t  Density: ρ  Pressure: p  Reynolds Number: Re

Velocity Components: (u,v,w)  Stress: τ  Heat Flux: q  Prandtl Number: Pr

**Continuity:**
$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0$$

**X – Momentum:**
$$\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho u^2)}{\partial x}+\frac{\partial(\rho uv)}{\partial y}+\frac{\partial(\rho uw)}{\partial z}=-\frac{\partial p}{\partial x}+\frac{1}{Re_r}\left[\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{xy}}{\partial y}+\frac{\partial \tau_{xz}}{\partial z}\right]$$

**Y – Momentum:**
$$\frac{\partial(\rho v)}{\partial t}+\frac{\partial(\rho uv)}{\partial x}+\frac{\partial(\rho v^2)}{\partial y}+\frac{\partial(\rho vw)}{\partial z}=-\frac{\partial p}{\partial y}+\frac{1}{Re_r}\left[\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{yz}}{\partial z}\right]$$

**Z – Momentum:**
$$\frac{\partial(\rho w)}{\partial t}+\frac{\partial(\rho uw)}{\partial x}+\frac{\partial(\rho vw)}{\partial y}+\frac{\partial(\rho w^2)}{\partial z}=-\frac{\partial p}{\partial z}+\frac{1}{Re_r}\left[\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}\right]$$

**Total Energy – Et:**
$$\frac{\partial(E_T)}{\partial t}+\frac{\partial(uE_T)}{\partial x}+\frac{\partial(vE_T)}{\partial y}+\frac{\partial(wE_T)}{\partial z}=-\frac{\partial(up)}{\partial x}-\frac{\partial(vp)}{\partial y}-\frac{\partial(wp)}{\partial z}$$
$$+\frac{1}{Re_r}\left[\frac{\partial}{\partial x}(u\tau_{xx}+v\tau_{xy}+w\tau_{xz})+\frac{\partial}{\partial y}(u\tau_{xy}+v\tau_{yy}+w\tau_{yz})+\frac{\partial}{\partial z}(u\tau_{xz}+v\tau_{yz}+w\tau_{zz})\right]$$
$$-\frac{1}{Re_r Pr_r}\left[\frac{\partial q_x}{\partial x}+\frac{\partial q_y}{\partial y}+\frac{\partial q_z}{\partial z}\right]$$

( 2.2.1)

Any thermal problem including boundary condition is feeded to software in terms of partial differential equation. this partial differential equation may not may not be solved by analytical methods, however in case of a general boundary condition, it is not possible to obtain analytical solution, therefore industrial software uses different numerical methods to give most accurate result of given problem, where the basis of any numerical methods is same and that is to discretize a system with infinite degrees of freedom into a finite system and to find most accurate result possible. Any thermal software calculates the thermal distribution and related thermal quantities in a system or component.

Typical thermal quantities of interest are:

- The temperature distribution
- The amount of heat lost or gained
- Thermal gradients
- Thermal fluxes

Some of the commonly used numerical methods / techniques include

- Finite element method (FEM)
- Boundary element method (BEM)
- Discrete element method (DEM)
- Finite volume method (FVM)
- Finite difference method (FDM)

So, these are all different techniques with different mathematical foundations the first two are more commonly used in structural-related problems; the third four problems that involve particulate behavior; the last two are commonly used in fluid-flow related problems.

## 2.2 Numerical methods

**Finite Element Method:**

Finite Element Method (FEM), also known as Finite Element Analysis (FEA) is a specific numerical technique that solves a continuous problem stated in the form of a PDE, by discretizing the problem into a finite number of nodal points. FEM works best in structural problems. Although FEM is widely used and successful numerical technique, cannot be used in every problem, for ex:- Fluid-flow

Problems related to fluid flow are described by Navier-Stokes equations. Due to the presence of advection-convection terms (or first order terms) in this problem, FEM does not perform well in certain conditions.

**Finite Difference Method:**

Finite difference methods describe the unknown's of the flow problem by means of point samples at the node points of a grid co-ordinate lines. Truncated Taylor series expansion is often used to generate finite difference approximations of derivatives of in terms of the point samples at each grid point and its immediate neighbors. Those derivatives appearing in the governing equations are replaced by finite differences yielding an algebraic equation for the values of at each grid point the order of accuracy depends on the highest order of the Taylor series expansion terms that eliminated.

**Finite Element Method:**

- The finite volume method was originally developed as a special finite difference formulation. It is central to four of five main commercially available CFD codes: PHOENICS, FLUENT, FLOW3D AND STAR-CD. The numerical algorithm consists of the following steps.
- Formal integration of the governing equations of fluid flow over all the (finite) control volumes of the solution domain.
- Discretisation involves the substitution of a variety of finite difference type approximations for the terms in the integrated governing equation representing flow processes such as convection, diffusion and sources. This converts the integral equation into system of algebraic equation.
- Solution of the algebraic equations by an iterative method.

The first step, the control volume integration, distinguishing the finite volume method from all other techniques. The resulting statements express the (exact) conversation of relevant properties for each finite size cell. This clear relationship between the numerical algorithm and the underlying physical principle forms one of the main attractions of finite element method and makes its concept much simple to understand by the engineers than finite volume method and spectral methods. The conservation of a general flow variable, for example a velocity component or enthalpy, within a finite control volume can be expressed as a balance between the various processes tending to increase or decrease it.CFD codes contain discretizations techniques suitable for the treatment of the key transport phenomena, convection (transport due to fluid flow) and diffusion (transport due to variations of from point to point) as well as for the source terms.

Addition to this thermal analysis can be classified into two ways:

**Steady state analysis:**

steady state thermal analysis used to determine heat flow rate, temperature, thermal gradients, heat fluxes in a object that are caused by thermal load that do not vary over time . A steady state thermal analysis calculates the effects on component; It is considered a good practice to perform steady state analysis before transient analysis.

**Transient analysis:**

Here thermal load as well as above mention quantities varies over time, thermal analysis calculates these quantities after every short time interval. This analysis most used to find creep of mechanical components

# CHAPTER 3

# Thermal Modeling in Ansys

Ansys is one of the widely used simulation software that offers different module different thermal. Initially for thermal analysis ansys AIM module was used Since it is very user friendly it is easy to create geometry , meshing (very accurate) , but particularly for heatsinks physics setup and boundary condition becomes quite complex at times and solution of simulation hardly satisfies our needs.

After following many tutorials ansys icepak turned out to be the best option which is special designed for electronic circuits. Just to ensure the accuracy of given module circuit check containing heatsink, source and fan is generated and meshed.

As per boundary condition following quantities are obtained:

- velocity in whole cabinate
- pressure at nodes
- 3D temperature profiles

## 3.1 Importing a cad model of heatsink

Though ansys provides its 3D modeling software as Design Modeler, generating complex geometry in design modeler is not user-friendly compared to other modeling software such as creo, solidworks, NX etc. Design modeler supports cad model generated in any above mentioned software and can be imported. After certain modification model is then transferred into icepack through workbench which gives access to collaborate between more than one software. Here icepak automatically configures all the faces, nodes, projection of cad model and surrounds a cabinet .As design of heatsinks are meant for increasing the surface area generally it comes with surression on its faces

Selected heatsink geometry model consist of :

**Heatsink:**

Length: 87

Width: 100

Height: 80

Surface area: 2630 $mm^2$

Further according to requirement external modules such as heat source, fan, and grill wall are merged in model

**Fan:**

Icepak offers three types of fan intake, exhaust and internal fan, here intake fan kept on one of the wall of cabinet.

Length: 80mm

Width: 80mm

Height: 25mm

Bearing: sleeve bearing

**Source:**

Length: 21.7mm

Width: 3mm

Height: 57.4mm

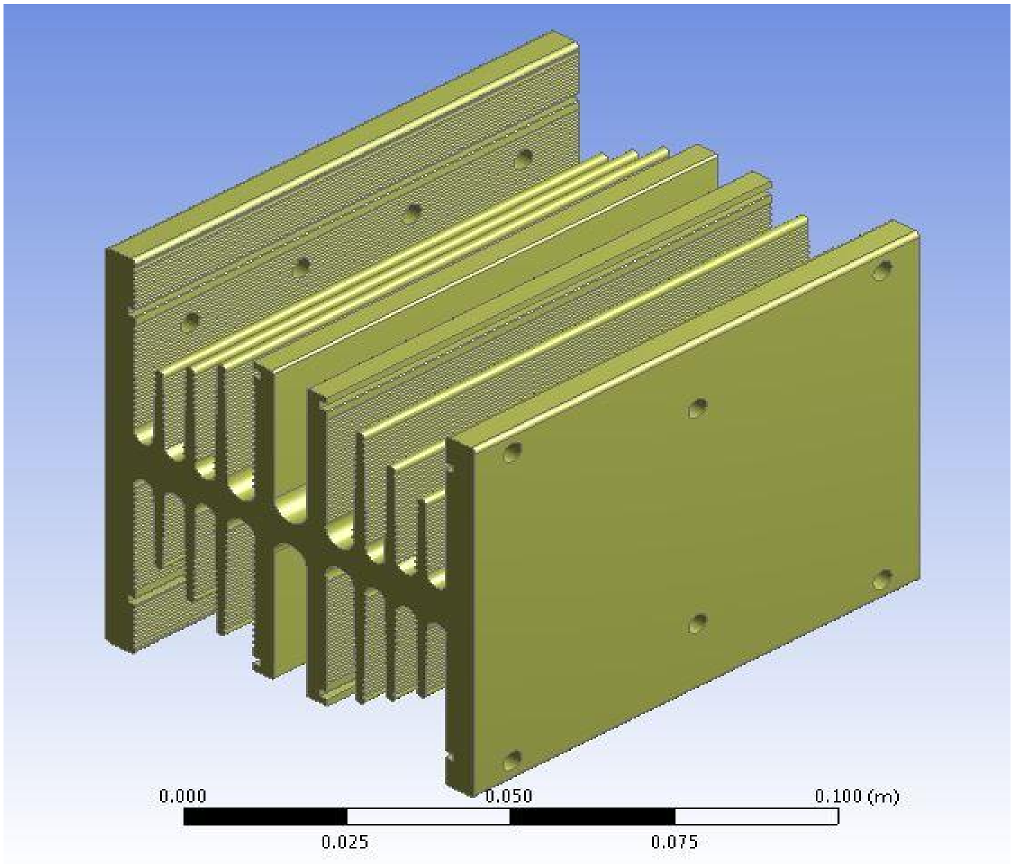



**Figure 3 Heatsink model**

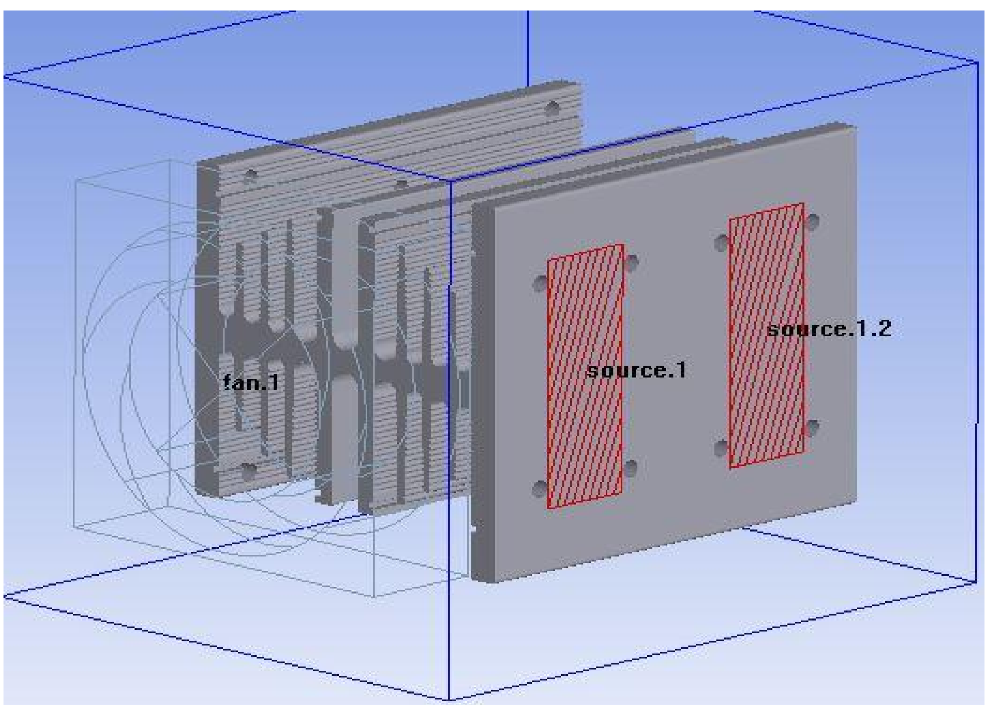



**Figure 4 Heatsink model with addins**

## 3.2 Meshing

This is the most important step of simulation, which decides the accuracy of solution. Meshing principally means to calculate number of nodes (points) and its location on whole body of a cad model. These finite numbers of elements are exploited by the solver too calculate solution. At every point software calculates parameters such as temperature, velocity, pressure etc.

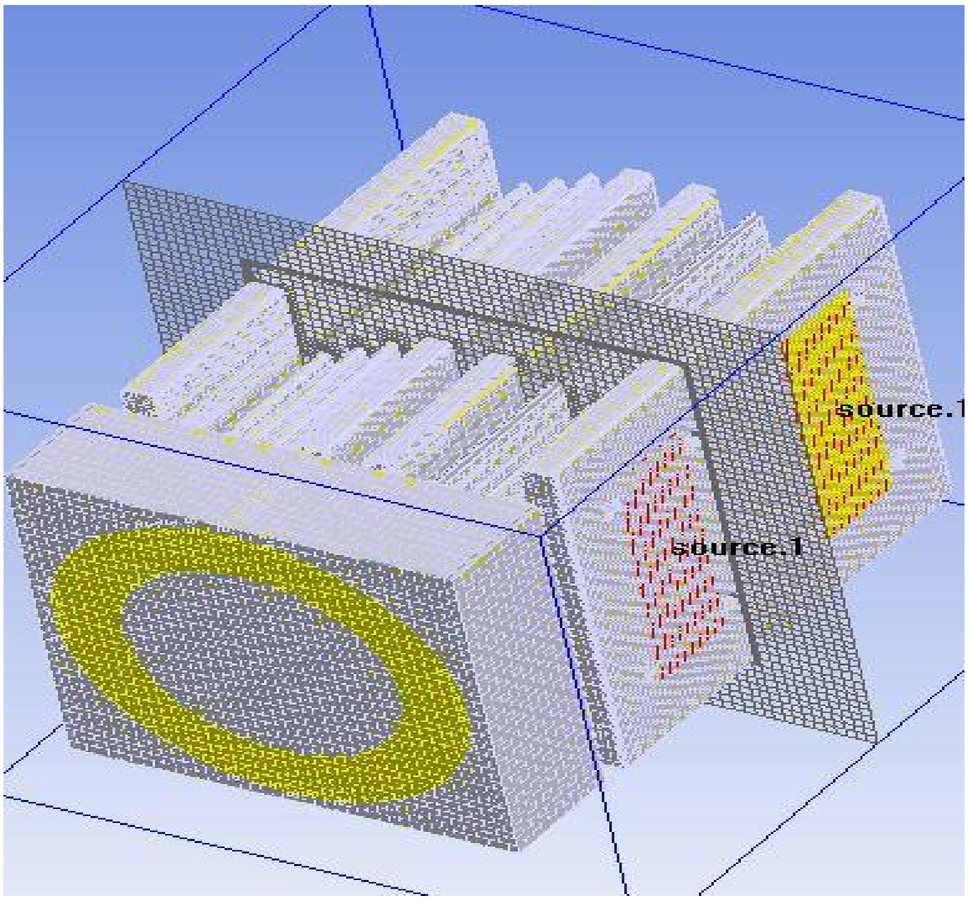



**Figure 5 Meshing of whole model**

Calculation of nodes are carried out either automatically or by the user specified parameters , in industry heatsinks have complex geometry therefore proper care should be taken while assigning meshing parameters, Choosing wrong parameters may cause crashing of solver.

The mesh setting parameters can be assigned in two ways:

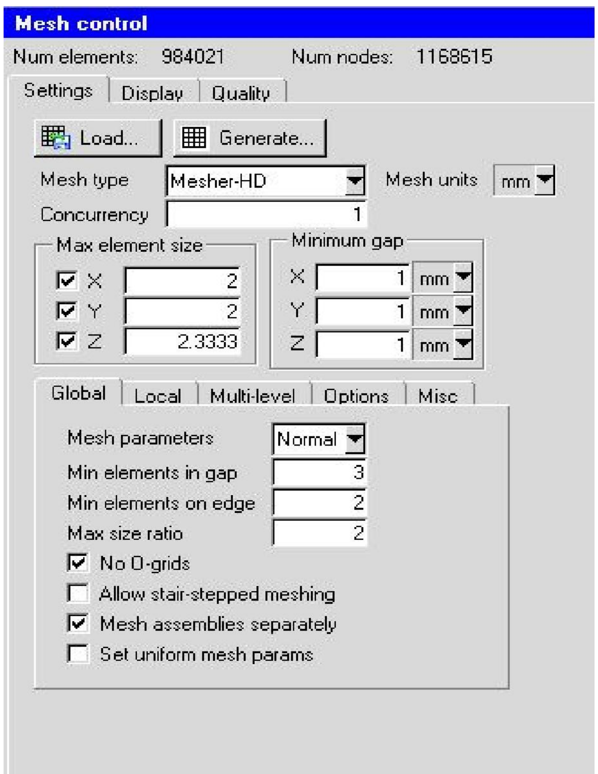
### User specified

MesherHD mesh type was selected to generate finite element mesh in entire volume of the model.

### Software specified

Mesh is specified in approximate manner, by selecting fine, normal, coarse mesh.

In this heatsink model meshing was in user specified way, the highlights of the settings are:



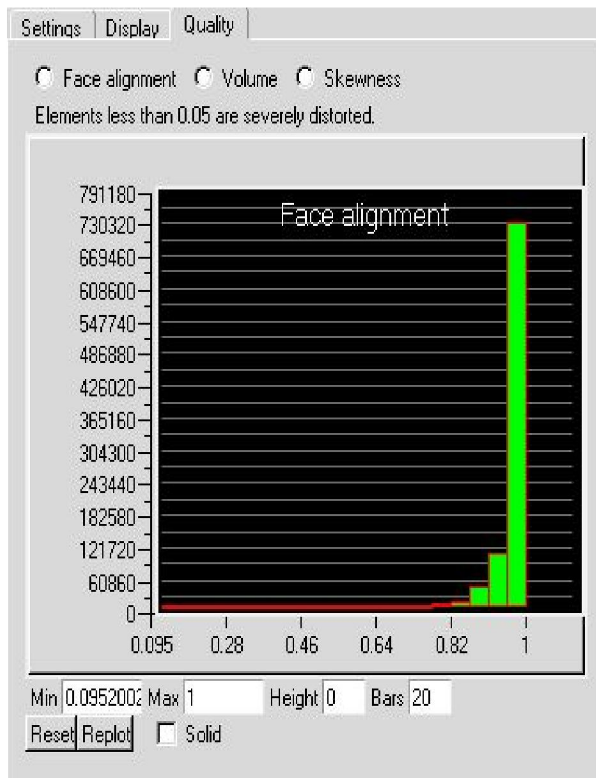




**Figure 6 Meshing parameters**

**Figure 7 Meshing parameters**

## 3.3 Setup of thermal and boundary condition

This step contains all the input essential for the thermal analysis

**Heatsink**:

Various materials such as metal , plastic ,alloys can be assigned using icepak libraries, Library Materials itself hold all thermal property such as thermal conductivity, thermal resistance coefficient and other flow property which may be changed by user , such material properties may be linear or non linear(vary with temperature)

**Material**: aluminum 6063 Thermal resistance: .65 $\frac{℃}{W}$

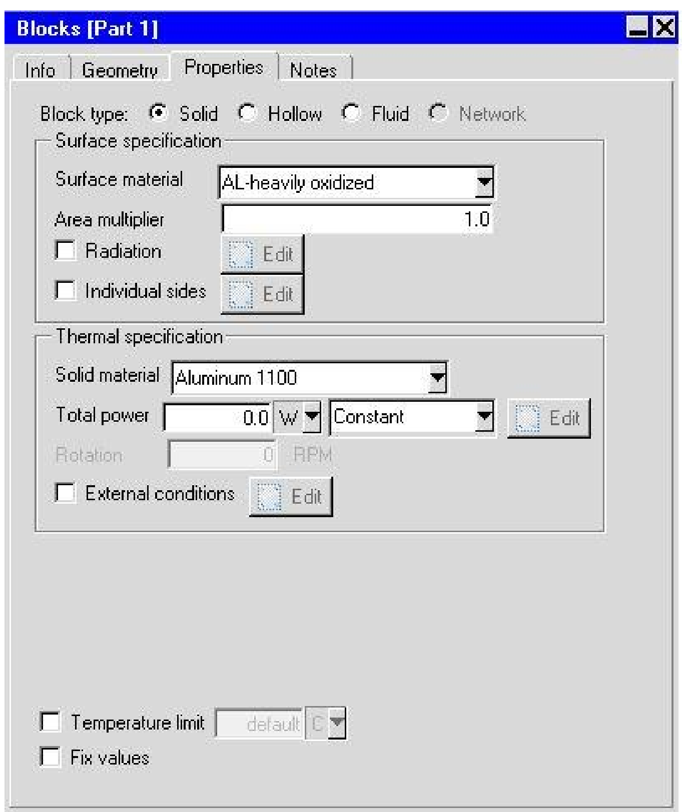



Figure 8 Assigning material

### Heat source:

Heat input is given either in terms of fixed temperature or input power

Fixed temperature: 125°C

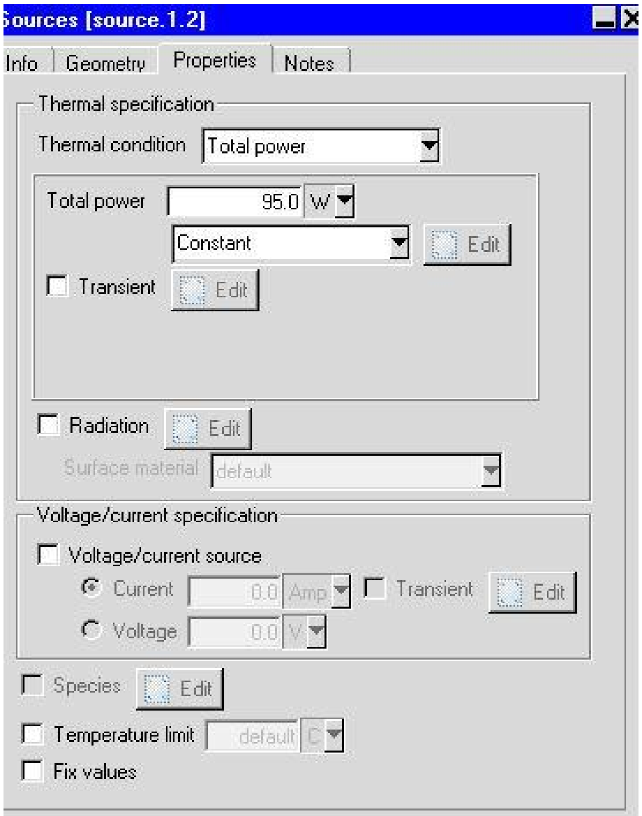



**Figure 9 Input power**

### Fan:

Volumetric flow: 360 cfm

Rpm: 2400

More to this surrounding condition such as velocity of fluid, ambient temperature operating pressure is set to the default value to by software and may be assigned by user

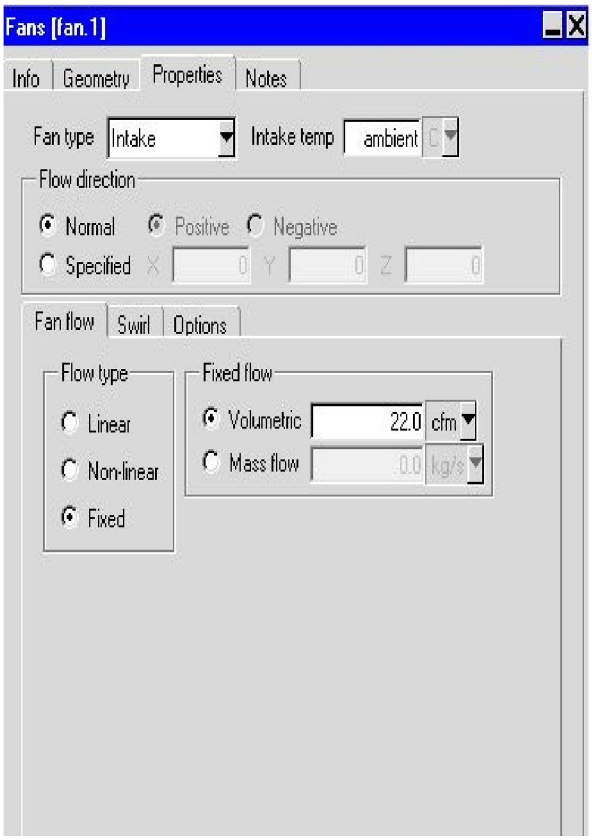



Figure 10 Fan input parameters

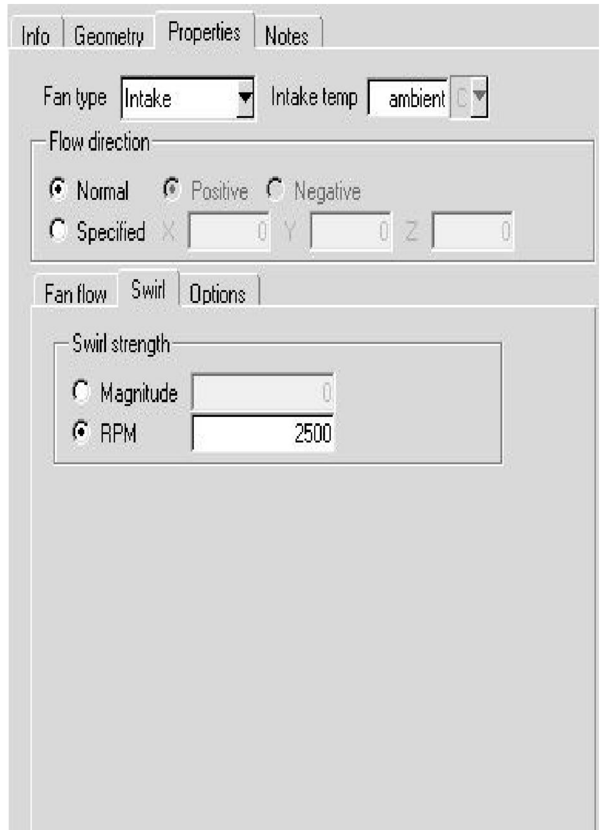



Figure 11 Fan input parameters

## 3.4 Solver setup

### Flow diagram:

The whole solver setup can be best elucidated through diagram shown below:

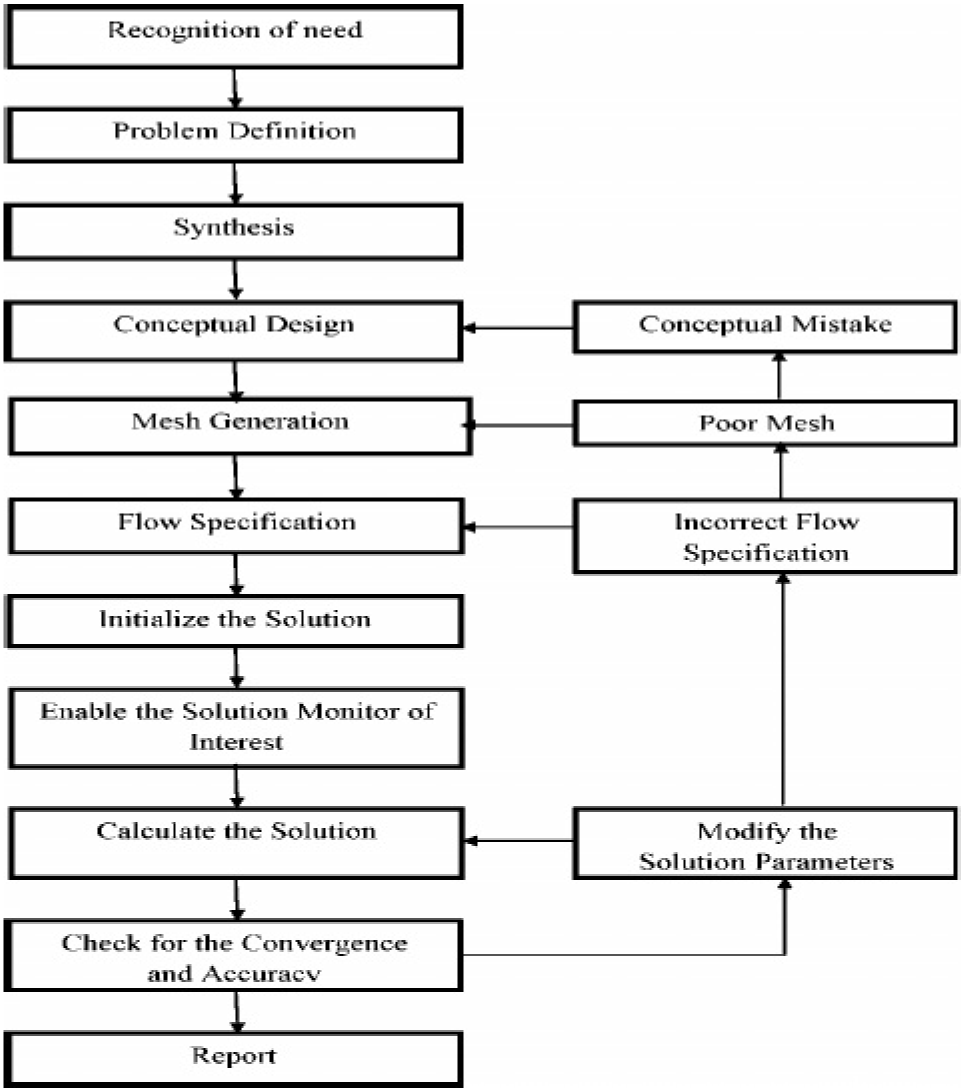



Figure 12 Process flow in ansys

As mentioned before ansys icepak uses FLUENT as solver, which uses Finite Volume Method apart from this as user certain choices are to be done for accurate solution:

**Flow regime**: turbulent

**Time variation**: transient

**Precision**: double

**Iteration per time step**: 10

Summary of this settings are:

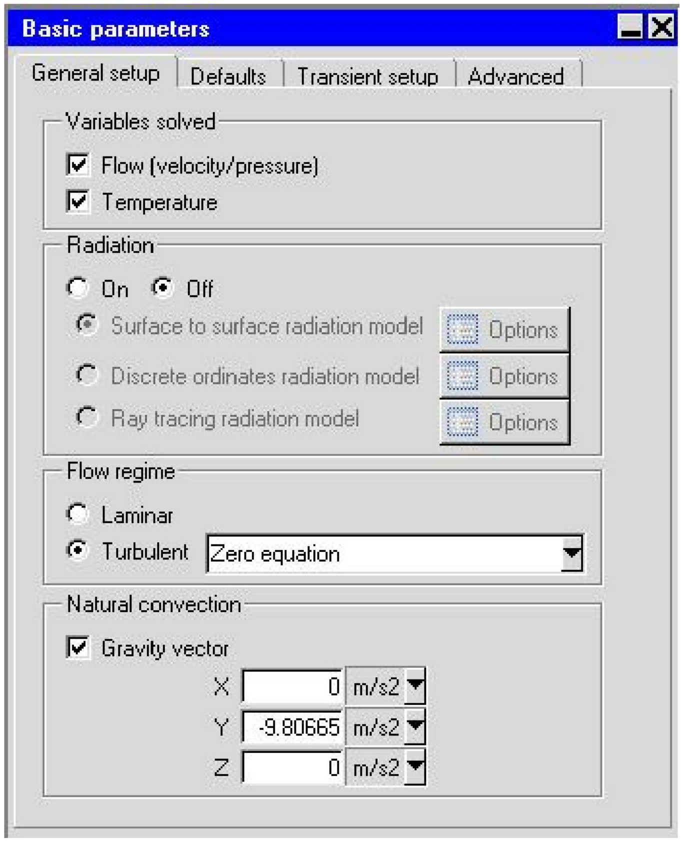
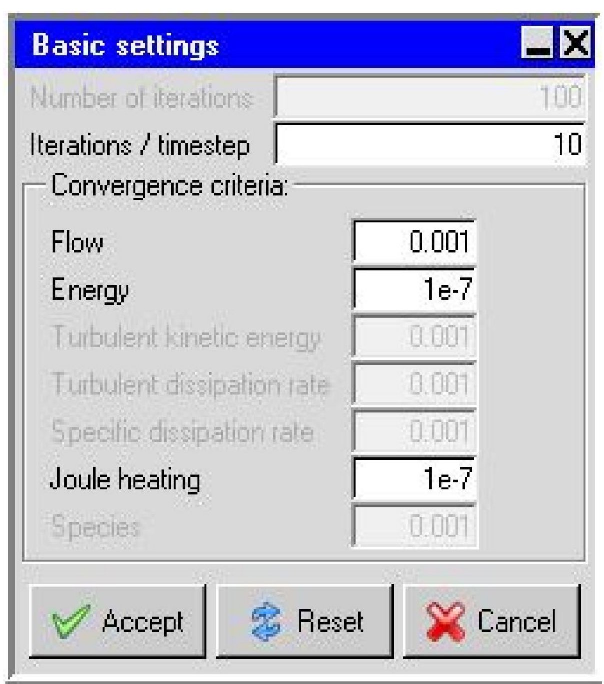



**Figure 13 general setup**



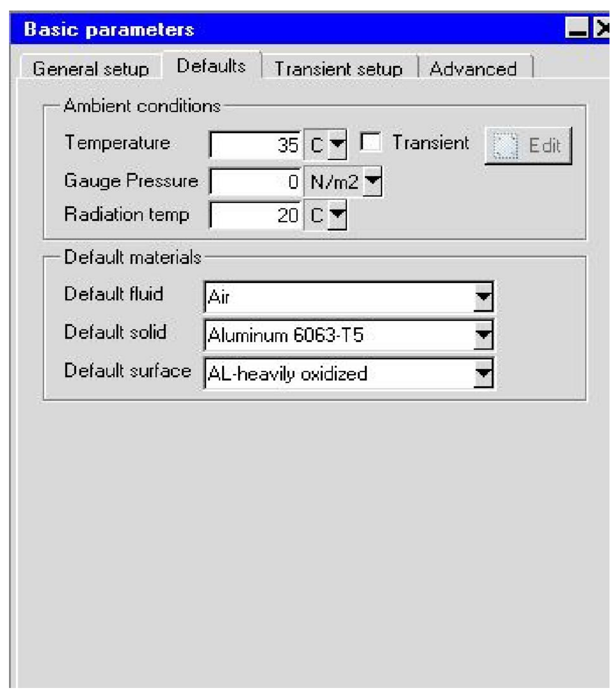




**Figure 14 basic setting**

**Figure 15 Default setup**

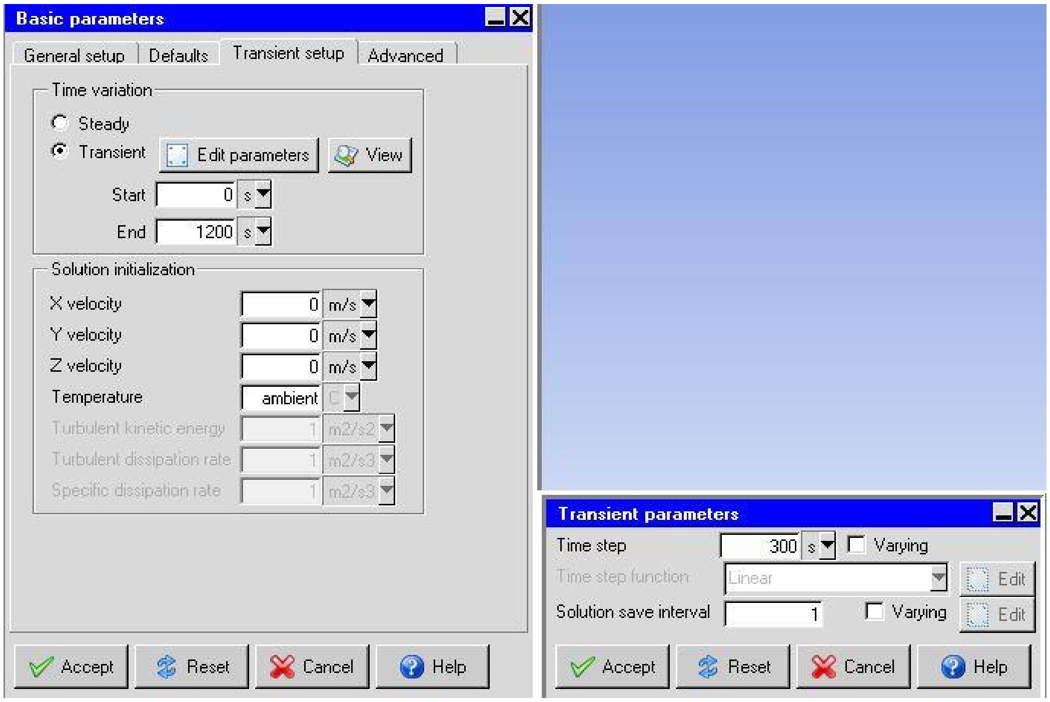



Figure 16 Transient setting

## 3.5 Post processing

As in pre-processing a huge amount of development work has recently taken place in the post-processing field. Owing to the increased popularity of engineering workstation, many of which have outstanding graphics capabilities, the leading CFD packages are now equipped with versatile data visualization tools.

This includes:

- Domain geometry and grid display
- Vector plots
- Line and shaded contour plots
- 2D and 3D surface plots
- Particle tracking
- View manipulation (translation, rotation, scaling etc)
- Color postscript output

More recently these facilities may also include animation for dynamic result display and in addition to graphics all codes produce trusty alphanumeric output and have data export facilities for further manipulation external to the code. As in many other branches of CAE the graphics output capabilities of CFD codes have revolutionized the communication of ideas to non specialist.

Here Ansys Icepak offers post processing option which is equipped with above mentioned visualization tools, apart from this Ansys offers separate CFD Post Processing module which specially designed for the same purpose.

## Icepak post processing

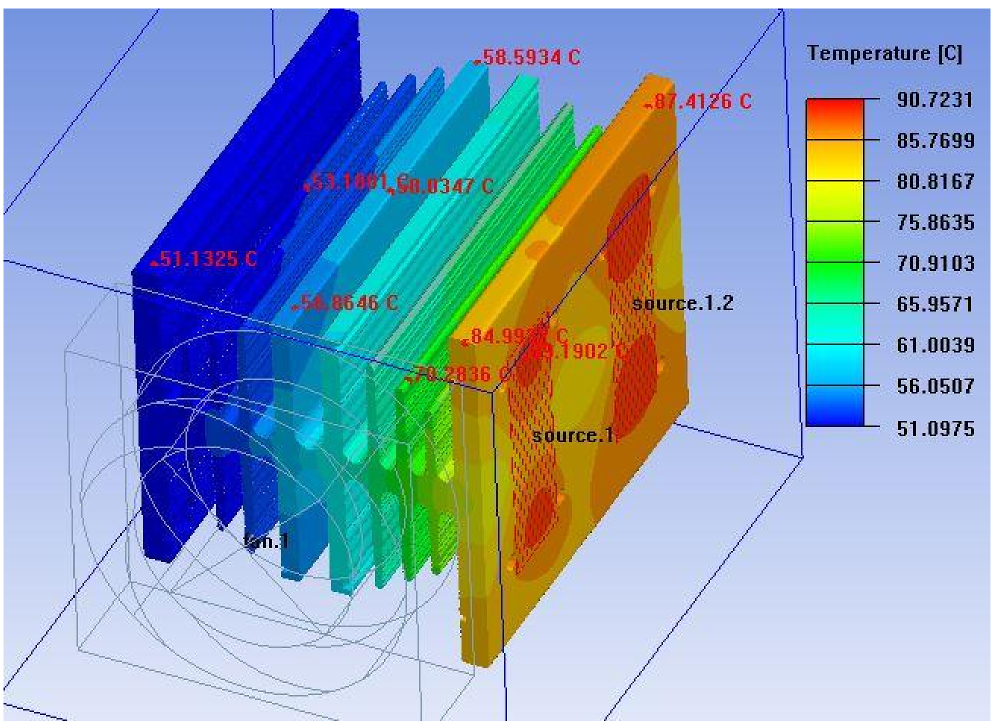



**Figure 17 Contour display: variable temperature with point mapping**

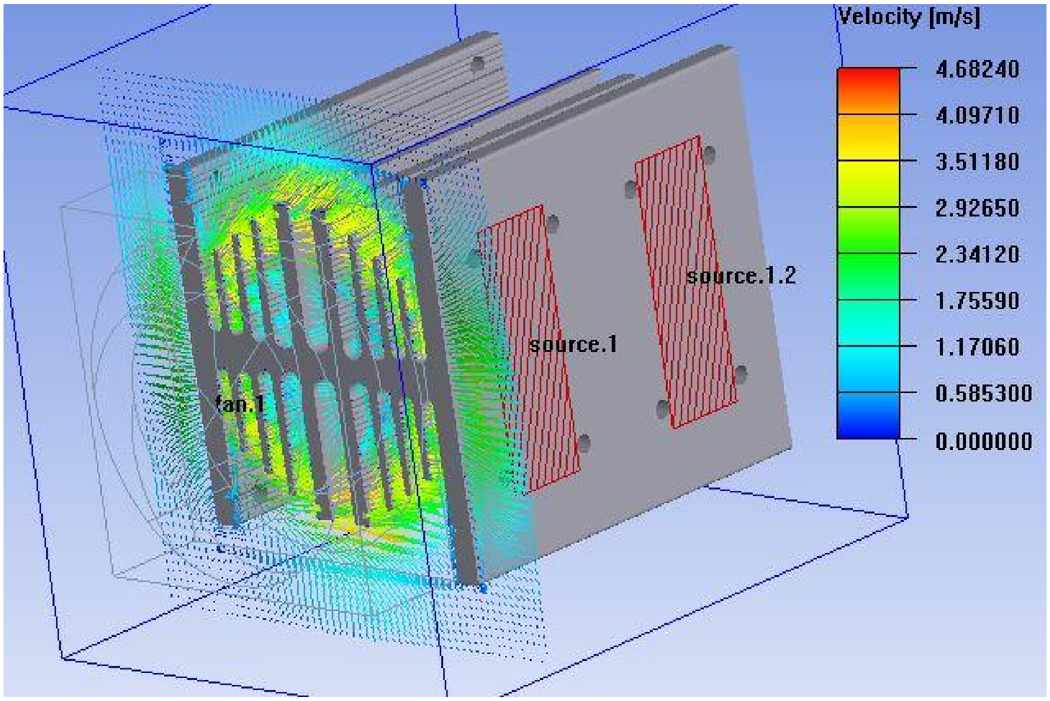



Figure 18 Plane cut: velocity

## CFD post processing

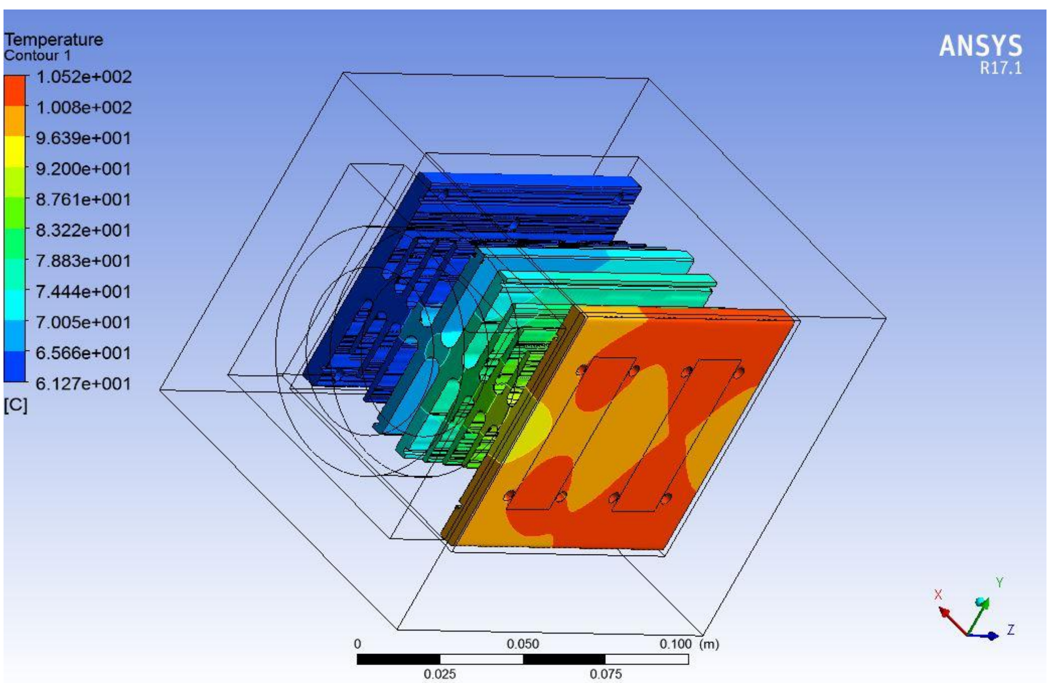



Figure 19 Time step 1

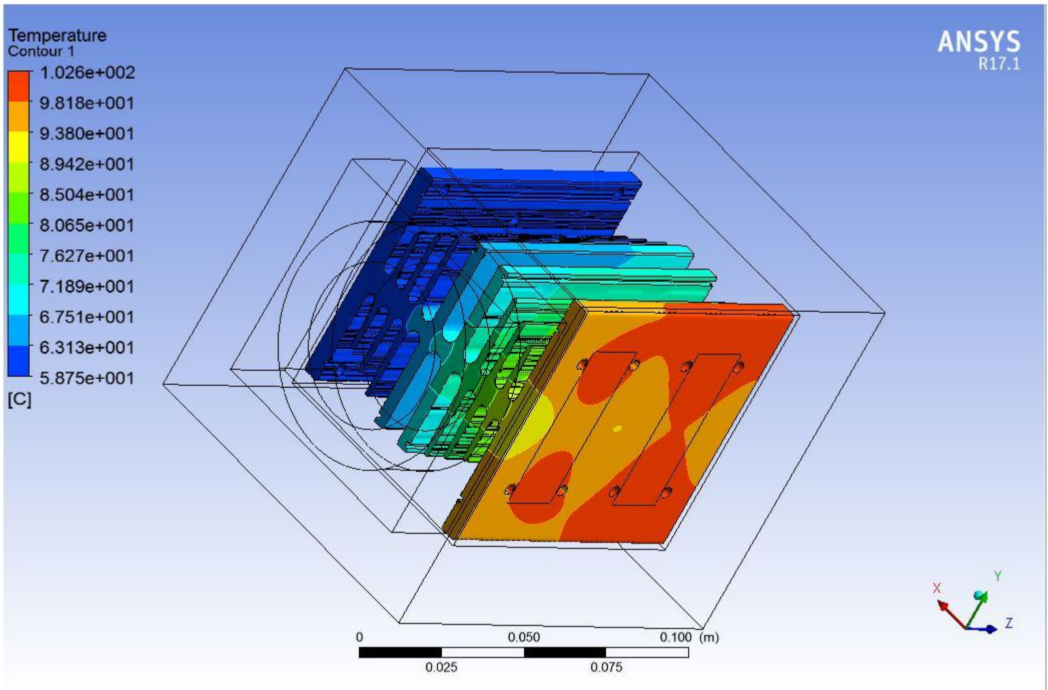



**Figure 20 Time step 2**

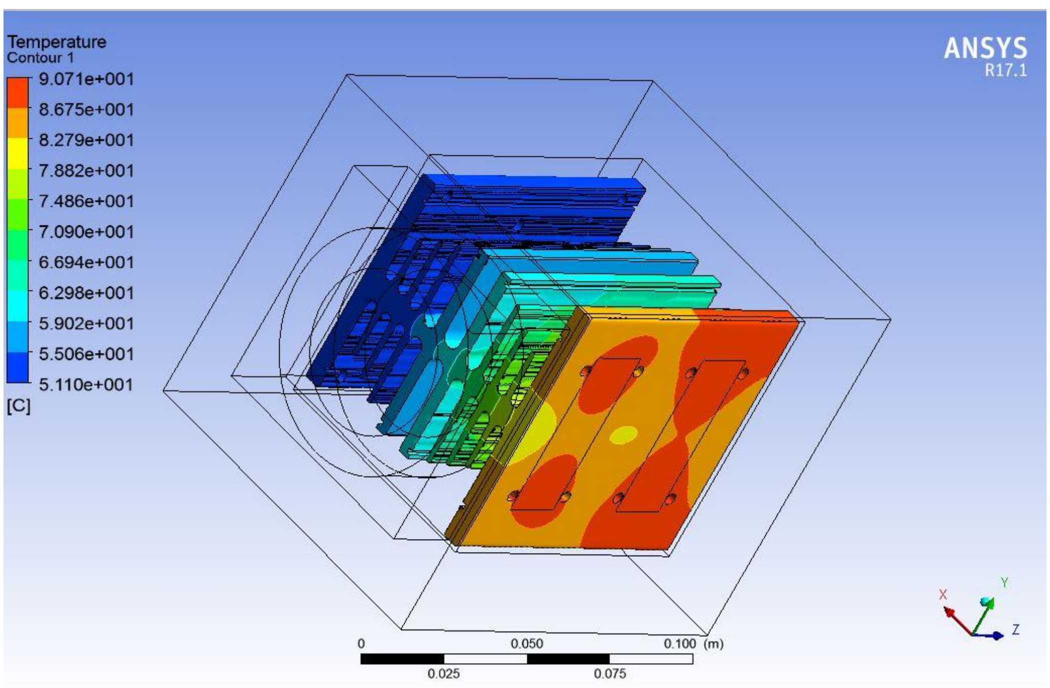



**Figure 21 Time step 3**

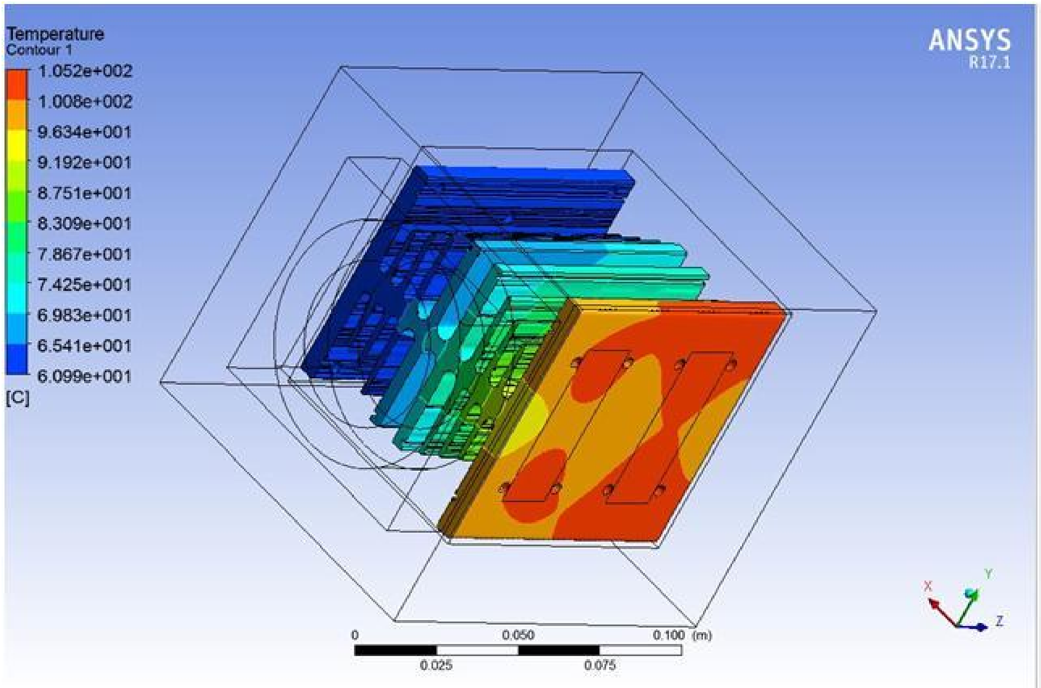



**Figure 22 Time step 4**

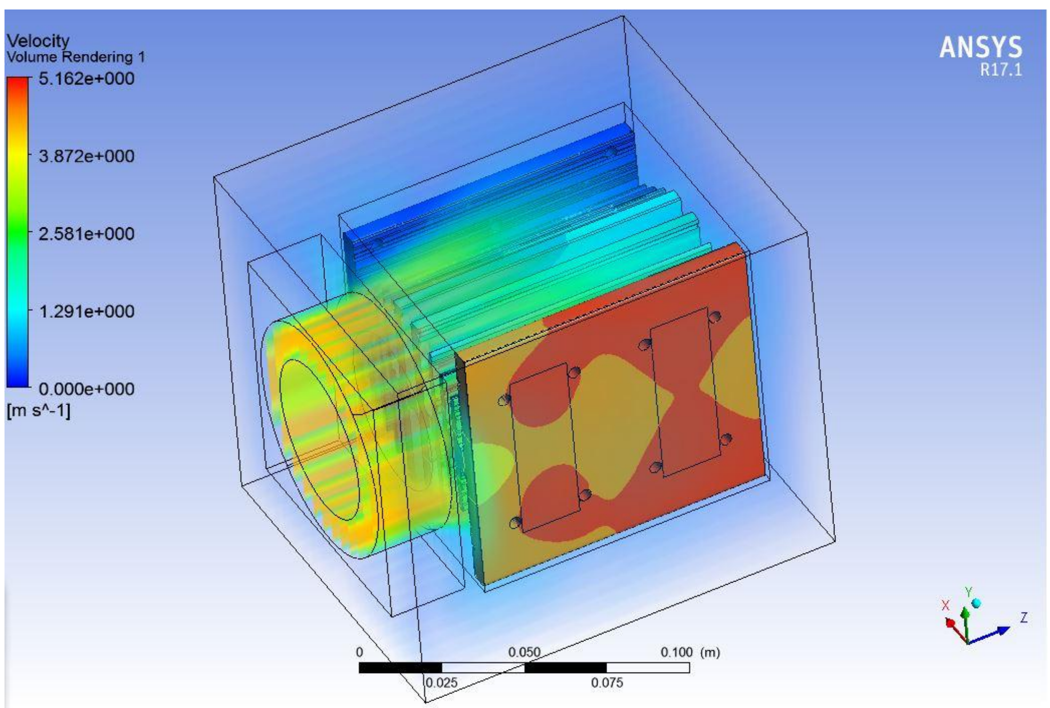



**Figure 23 Volume rendering: velocity**