Chapter 5

NUMERICAL ANALYSIS

5.1 GEOMETRIC MODELLING

The first step in the CFD analysis was the geometric modelling of the micro channel assembly as a three layer one. The layer in the middle represented the liquid coolant flow and the adjacent layers represented the solid walls of the channels. The geometric parameters of the micro channel were modelled, as a width of 150 μ and a height of 7000 μ resulting in an aspect ratio of 46.66, with a length of 31 mm, in ICEMCFD software. The solid wall of 75 μ was modelled on both sides of the coolant steam.

The second stage of modelling was the meshing of the geometry. A check for consistency and surface errors was done on the 3D model of the micro channel before meshing it the next stage as the simulation part which was done on CFD simulation software ANSYS CFX 14.5. The liquid coolant used in our numerical study for validation and parametric studies was water, which was also used as a coolant in our experimental work. The continuum flow through the micro channel was assumed inside the micro channel heat exchanger and the flow field and heat transfer behaviour through the Micro Channel Heat Sinks was obtained by solving continuity equation and the Navier-Stokes equations. Care was taken to mesh the fluid domain and solid domain of the Micro Channel Heat Sinks separately. Also a proper interface was defined in ANSYS-CFX Pre for catching the flow physics. The meshing was performed in ICEMCFD tool using blocking method. Since it was not considered feasible to simulate the entire number of 122 channels, it was decided that one fluid domain and two solid domains, each half on either side of the fluid be simulated.

Symmetric boundary conditions were applied on either sides of the solid domain, since the Micro Channel Heat Sinks surfaces were rectangular in geometry. The numerically worked out results for 7 mm were validated with the experimental work, which was performed at NIT, Tiruchirapalli. There existed good agreement
between the numerical and the experimental results. The outcome of the current research study has similar results of Jung Yeul et al (2008) and they have reported that the Nusselt number increases with increase in Reynolds number for laminar flow while using water as coolant.

Hence, it was decided to perform numerical analysis on Micro Channel Heat Sinks models of varying heights of 3mm, 4.5 mm and 7 mm, while keeping the length and breadth of the micro channels constant with the same boundary conditions. The heights of 3, 4.5 and 7 mm corresponded to aspect ratios of 20, 30 and 46.66 respectively for the Micro Channel Heat Sinks. Also the thickness of the wall was kept constant at 150 μm for all the three cases. Actually the reason behind taking 20,30 and 46.66 AR micro channels was to establish that higher aspect ratio micro channels would give higher rate of heat transfers. Therefore, keeping the above in mind, the sizes were fixed at 20,30 and 40 but the fabrication difficulty in getting the size 40 aspect ratio size resulted in a size of 46.66. This was the nearest size to aspect Ratio of 40 that we could manufacture, and as a numerical comparison it was decided to hone in 30 and 40 as a means of comparison to get meaningful results in the Nusselt number, heat removal rates, friction factor etc.

The values are arrived at keeping the height 7 mm and a width of 150 μm for each rectangular micro channel used in the experimental work resulting in an aspect ratio of 46.66. Consequently, the height was reduced to 4.5 mm and 3 mm and resulted in aspect ratio of 30 and 20 respectively for numerical analysis and subsequent comparison with the experimental size.

The objective of creating additional geometries was to obtain an optimal size of Micro Channel Heat Sinks with higher heat transfer characteristics accompanied by minimal pressure drop. This numerical work was carried out in the CFD simulation lab of VIT University, Vellore.
5.2 GRID GENERATION-FINITE VOLUME DISCRETIZATION

The three-dimensional discretized model is created in the meshing tool ICEMCFD in the initial pre-processor phase. The model is completely discretized as finite volume hexahedral elements, which help capture the hydrodynamic and thermal boundary layers. In order to achieve the above, a y+ value of less than 1 has to be maintained near the fluid solid interface. In order to maintain the maximum quality while discretizing the domain, the minimum angle and determinant are maintained at 90 degree and 1 respectively. The micro channel has discrete zones differentiating the solid and liquid domains.

The fluid domain is surrounded by the solid copper wall domain all around it and the fluid—solid interfaces are defined in the pre-processor and described in Table 5.1. The fluid domain has Fluid Inlet and Fluid Outlet as velocity boundary inlet with pressure boundary at the exit. The interfaces clearly indicate how the fluid flows through the micro channels. The entire process of heat transfer through the Micro Channel Heat Sinks is conjugate heat transfer, which means that conduction of heat takes place through the wall, convection of fluid happens through the liquid. Also conduction takes place from the heaters to the bottom walls. Hence in order to take care of the coupling of conduction and convection effects for the heat transfer through the Micro Channel Heat Sinks, the meshing is constructed very fine. Radiation effects though present are minimal and were not considered in the numerical analysis.

<table>
<thead>
<tr>
<th>SL No.</th>
<th>Solid Domain</th>
<th>Fluid Domain</th>
<th>Interface/Boundary Condition</th>
<th>Interface/Boundary type</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Copper</td>
<td>Water</td>
<td>FLSO_Left</td>
<td>Fluid Solid</td>
</tr>
<tr>
<td>2.</td>
<td>Copper</td>
<td>Water</td>
<td>FLSO_Right</td>
<td>Fluid Solid</td>
</tr>
<tr>
<td>3.</td>
<td>Copper</td>
<td>Water</td>
<td>FLSO_Bottom Bottom Wall</td>
<td>Constant Heat Flux</td>
</tr>
<tr>
<td>4.</td>
<td>Copper</td>
<td>---</td>
<td>Bottom Wall</td>
<td>Constant Heat Flux</td>
</tr>
<tr>
<td>5.</td>
<td>Copper</td>
<td>Water</td>
<td>FLSO_Top Top Wall</td>
<td>Adiabatic</td>
</tr>
<tr>
<td>6.</td>
<td>Copper</td>
<td>---</td>
<td>Fluid_Inlet</td>
<td>Adiabatic</td>
</tr>
<tr>
<td>7.</td>
<td>---</td>
<td>Water</td>
<td>Fuid_Outlet</td>
<td>Velocity Inlet</td>
</tr>
<tr>
<td>8.</td>
<td>---</td>
<td>Water</td>
<td>Fuid_Outlet</td>
<td>PressureOutlet (0 Gauge pressure)</td>
</tr>
</tbody>
</table>

Table 5.1 Boundary Conditions and Interfaces evolved in the ANSYS-CFX Processor
5.3 GOVERNING EQUATIONS

The governing equations are the conservation of mass viz. the continuity equation, conservation of momentum and energy viz. the Navier-Stokes equations for predicting the heat transfer.

Assumptions which were used in our analysis to write the above differential equations were:

1. Steady fluid flow and heat transfer who means that there is streamline flow and transient heat phenomenon.
2. The liquid, used as the coolant, is assumed to be incompressible flow, which means that its density is assumed constant.
3. Laminar flow assumption indicates steady state flow.
4. The constant solid and fluid flow properties assumed indicate that there no measurable changes in the same.

The above assumptions helped us write the governing differential equations, for fluid flow and heat transfer, in the following manner.

Continuity Equation:

$$\nabla \cdot (\rho \vec{V}) = 0$$  \hspace{1cm} (5.1)

Momentum Equation:

x-momentum: $\nabla \cdot (\rho u \vec{V}) = \frac{\partial p}{\partial y} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z}$ \hspace{1cm} (5.2)

y-momentum: $\nabla \cdot (\rho u \vec{V}) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho g$ \hspace{1cm} (5.3)

z-momentum: $\nabla \cdot (\rho u \vec{V}) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z}$ \hspace{1cm} (5.4)

Energy Equation:

$$\nabla \cdot (\rho e \vec{V}) = -p \nabla \cdot \vec{V} + q + \phi$$ \hspace{1cm} (5.5)
The equations are solved with the help of a segregated solver. This means that the continuity and momentum equations are solved in sequence i.e. one after the other. Since the governing equations are non-linear, much iteration are done in a closed loop inside the solver to get a convergent solution. Momentum equations are solved using the Navier-Stokes equations. Here the average velocity components are considered and since the flow is laminar, turbulence is not considered in the micro channels.

The equations of continuum are valid for our Micro Channel Heat Sinks model. In order to model the laminar fluid flow with viscous friction, upwind scheme is used. The Micro Channel Heat Sinks wall surfaces are smooth and hence there is no slip condition at the wall surface. Since the flow velocities are quite low, laminar flow is considered in our numerical analysis.

5.4 NUMERICAL SOLUTIONS

The finite volume method has been used to solve the governing equations of continuity, momentum equations. In the above method, the domain is divided into many finite control volumes with each control volume surrounding one grid point. The governing equations are then integrated throughout the control volume to get algebraic equations for the discretized pressure, temperature and velocities.

The solution for the above equations is assumed to converge when the mass and momentum imbalance is less than $10^{-6}$ and for energy it is less than $10^{-8}$. This means that in the solver, by repeated iterative runs of the equations, the errors converge to a minimum of five decimal places for mass and momentum and up to seven decimal places for energy which is nearly zero. Hence the best solutions would have higher values of convergence.

5.5 BOUNDARY CONDITIONS

The inputs for the model are the boundary conditions defined in the ANSYS-CFX processor. Table 5.1 mentions the boundary conditions. The fluid considered in our analysis was water with its thermal and physical properties considered at 33°C.
a) The inlet coolant temperature is 33°C.

b) A constant heat flux is assigned to the bottom surfaces.

c) The inlet is velocity and is assumed uniform.

d) No-slip boundary condition is assigned to all the wall surfaces.

e) The side and top walls of the Micro Channel Heat Sinks are made insulated.

Zero gauge pressure is assigned at the outlet of the Micro Channel Heat Sinks

5.6 POST PROCESSING AND DATA EXTRACTION

The post processor carries out the final operations for deducing the data in the ANSYS-CFX software after developing the model, discretizing and solving the governing equations with the appropriate boundary conditions.