CFD Simulation on Heat Transfer Characteristics of Laminar Air Flow On Dimple and Flat Plate
Shekhar Pandey\textsuperscript{a}, Radhesh Krishnan\textsuperscript{b}, Mithun Sarkar\textsuperscript{c}, Nirmal Singh\textsuperscript{d}, Gaikwad S.M.\textsuperscript{e}

\textsuperscript{a,b,c,d,e} Army Institute of Technology, Pune
\textsuperscript{a}shekhar8198@gmail.com, \textsuperscript{b}radhesh18@rediffmail.com

Abstract: This paper presents the numerical study performed with ANSYS FluidFlow (CFX) to compare heat transfer characteristics of a flat plate and an inline dimpled surface plate configuration which is placed on the wall of a rectangular duct. For this simulation, laminar model was used. A square mild steel plate was taken and it was enclosed in a duct from which the air was passed through at ambient temperature (20\textdegree{}C). The dimple depth to cavity curvature diameter ratio was taken as 0.25. Heat transfer characteristics and velocity profile were investigated using numerical techniques for dimple plate and the results were compared to that of the flat plate.

1 INTRODUCTION
Surface indentations or dimples are widely used in several industrial applications. Some of the examples are gas turbine cooling passages, heat exchangers, and flow separation control over the round cylinder or over the airfoil. In most flat plates, the flow is laminar due to small channel dimensions and low fluid velocities and as a result heat transfer coefficient is very low. One of the method to enhance this insufficient heat transfer rate is to provide indentations normal to the main flow. These indentations interrupt the hydrodynamic boundary layer periodically, add surface area, generate secondary flows and vortices, and increase flow velocity. The enhancement in heat transfer rate is extremely dependent on Reynolds number, free stream turbulence, the indentation geometry and configuration. Heat transfer in these dimpled surface is enhanced due to periodic interruptions of thermal boundary layers and also improvement in lateral mixing by disruption of the shear layer, separation of the bulk flow, formation of recirculating flows, and thus destabilization of the transversal vortices in the dimples.

2 NOMENCLATURE
The terms used in the paper with their SI units are given below:

\begin{align*}
D & [\text{m}] \quad \text{Cavity curvature diameter} \\
D_h & [\text{m}] \quad \text{Hydraulic diameter} \\
H & [\text{m}] \quad \text{Duct Height} \\
K & [\text{W m}^{-1} \text{K}^{-1}] \quad \text{Fluid thermal conductivity} \\
L & [\text{m}] \quad \text{Duct length} \\
Nu & [-] \quad \text{Nusselt number} \\
\delta & [\text{m}] \quad \text{Cavity depth} \\
P & [\text{Pa}] \quad \text{Fluid static pressure} \\
q_w & [\text{W m}^{-2}] \quad \text{Wall heat flux} \\
Re & [-] \quad \text{Reynolds number} \\
T_i & [\text{K}] \quad \text{Fluid temperature at inlet} \\
T_w & [\text{K}] \quad \text{Wall temperature}
\end{align*}

3 PHYSICAL MODEL
Two square plates of dimension 148X148 mm having thickness of 17 mm were taken. Each of them was enclosed in a rectangular duct one after another. The dimensionless length of the enclosure is considered as L=600 and the dimensionless height of the enclosure is considered as H=150. The Inline dimple configuration was provided on one of the square plate which was equally distributed in 3X3 matrix formation. The second plate was taken as a flat plate with no dimples. The projection length (or print diameter) of the cavities was 24 mm. The distance between adjacent dimples was 20 mm. The dimple depth, \( \delta \) was taken as 6 mm. The plates were heated to a temperature of 100\textdegree{}C isothermally and maintained at that temperature. Air at 20\textdegree{}C was introduced through an opening on the left vertical wall and exited through the other side of the duct. All walls were assumed to be adiabatic and dimple surface is considered as the isothermally heated. Thus, the
heat transfer process is done by mixed convection. Both the size of the inlet section is same as that of exit opening.

Figure 1: Geometry of flat plate surface. Figure 2: Geometry of dimple surface.

4 COMPUTATIONAL METHODOLOGY

The governing equations are the continuity, momentum, and energy equations. The flow is studied under the following assumptions: steady-state, constant fluid properties and no natural convection and body forces. The governing equations for steady mixed convection flow using conservation of mass, momentum and energy can be written as,

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0
\]

\[
u \frac{\partial u}{\partial x} + \nu \frac{\partial u}{\partial y} = -\frac{1}{2} \frac{\partial p}{\partial x} + \frac{1}{Re} \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right)
\]

\[
u \frac{\partial v}{\partial x} + \nu \frac{\partial v}{\partial y} = -\frac{1}{2} \frac{\partial p}{\partial y} + \frac{1}{Re} \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) + Ri \theta
\]

\[
u \frac{\partial \theta}{\partial x} + \nu \frac{\partial \theta}{\partial y} = \frac{1}{Re \cdot Pr} \left( \frac{\partial^2 \theta}{\partial x^2} + \frac{\partial^2 \theta}{\partial y^2} \right)
\]

The computational package, ANSYS CFX, was used for the numerical model. It is a 3D solver of the Reynolds Averaged Navier Stokes equations based on the finite volume formulation. Triangular cells are used to discretize the problem domain with a structured mesh. Grid points are distributed in a non-uniform manner with a higher concentration near the walls due to higher variable gradients expected in these locations. The convergence criterion is that the residual variations of the mass, momentum and energy conservation equations become less than 10^{-5}. The numerical model is validated by solving the velocity and temperature fields in dimple plate with smooth surfaces, constant inlet velocity and equal wall temperatures.

5. BOUNDARY CONDITIONS

5.1 Meshing:

Triangular element is used for the meshing of both the plate surface and duct. Further refinement of the dimple as well as flat plate surface has been done to enhance the flow of the air inside.
Table 1: Mesh Report

<table>
<thead>
<tr>
<th>Sr No.</th>
<th>Plate Configuration</th>
<th>Domain</th>
<th>Nodes</th>
<th>Element</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Flat</td>
<td>Fluidzone</td>
<td>60588</td>
<td>326790</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Plane</td>
<td>81954</td>
<td>342454</td>
</tr>
<tr>
<td>2.</td>
<td>Inline Dimples</td>
<td>Fluidzone</td>
<td>88893</td>
<td>476440</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Plate</td>
<td>115721</td>
<td>616901</td>
</tr>
</tbody>
</table>

5.2 **At all walls:** Since the Velocity Component, u and v both are zero at the walls for no-slip conditions, the walls are considered as adiabatic.

5.3 **At the plate surface:** Since the velocity component, u and v both are zero at the walls for no-slip conditions. The object is assumed to be isothermally heated.

5.4 **Boundary Condition at Inlet:**

**Boundary Condition at outlet:**

- Velocity of air = 0.1 m/s
- Static air Temperature = 20°C
- Pressure = 105 Pa

Expression

6. **RESULTS AND DISCUSSION**

To study the heat transfer characteristics of the flow in duct, the average values of wall heat flux and Nusselt number should be computed in each configuration. The average wall heat flux is computed by the following equation:

\[
\bar{q}_w = \frac{1}{L_{wall}} \int q_w(x) dx
\]

Where \( q_w(x) \) is the wall heat flux and is defined based per unit area of the wall and L is the total length of the duct. In a dimple plate with smooth surfaces, wall heat flux has a maximum value at the inlet and decreases along the wall in an exponential manner as the hydrodynamic and thermal boundary layers develop. Flow patterns and heat transfer characteristics have been studied numerically for flows over plate containing symmetrical arrays of spherical cavities.

6.1 **Temperature distribution**

Figure 3 shows the temperature variation of the air along the dimple after attaining steady state condition. It can be seen that the temperature of the air inside the dimple surface has more temperature than that of the air above. The maximum local heat flux happens at the downstream of each dimple which is due to large vorticities in these zones. As seen in the figure on the left, the temperature gradient along the Y-axis is greater as compared to that on the right due to formation of secondary flows and thicker boundary layer.
Figure 4 shows that moving along the length of the duct, initially temperature remains constant until the air stream strikes the dimple plate where it constantly rises and when the air stream hits the dimple the temperature drops suddenly. With the air stream moving past each dimple, temperature is dropped while absorbing a substantial amount of heat, air stream moving past the dimple plate, its temperature hikes.

6.2 Velocity distribution

Figure 5 shows the velocity vector of the air in the dimple surface. The velocity of the air is less in dimple. It is because of the formation of vortices inside the dimple surface.

Figure 6 shows that the air velocity inside the duct starts to decrease initially due to boundary layer formation and viscous effects. But as the air comes in contact with the dimple surface its velocity increases due to formation of vortices and turbulence in case of the dimple plate as can be observed in the figure on the above left. While on the flat plate the velocity is first decreased and after crossing the plate surface, the velocity increases marginally due to creation of the convective currents moving from the lower layers to the above layers.
Figure 5: Velocity vector plot for dimple plate and flat plate respectively.

Figure 6: Graph for Velocity variation along the duct for dimple plate and flat plate respectively.

Table 2: Results on CFD for different configuration of plates.

<table>
<thead>
<tr>
<th>Sr No</th>
<th>Parameter</th>
<th>Numerical Calculation (CFD)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Flat Plate</td>
</tr>
<tr>
<td>1</td>
<td>Reynolds’s Number</td>
<td>1207.300</td>
</tr>
<tr>
<td>2.</td>
<td>Prandtl Number</td>
<td>0.704620</td>
</tr>
<tr>
<td>3.</td>
<td>Nusselt’s Number</td>
<td>37.4268</td>
</tr>
<tr>
<td>4.</td>
<td>Wall Heat transfer Coefficient (W m⁻² K⁻¹)</td>
<td>25.88350</td>
</tr>
<tr>
<td>5.</td>
<td>Velocity at outlet (m s⁻¹)</td>
<td>0.099963</td>
</tr>
<tr>
<td>6.</td>
<td>Wall Heat flux (W m⁻²)</td>
<td>230.9210</td>
</tr>
</tbody>
</table>
7. CONCLUSION

The CFD heat transfer simulation over the spherical dimples has been carried out at the laminar flow conditions. Velocity and temperature fields are computed to study effect of the dimple surface on heat transfer characteristics. It can be seen that the Heat transfer coefficient of the dimple surfaced plate is more than that of the flat plate. Despite the dimple cavity enhances local heat transfer, the average heat transfer can be higher and lower of the flat plate data depending on the free stream velocity. This is due to the relatively low flow velocity and associated heat transfer near the dimple bottom. The results showed formation of vortices in cavities at the relative depth of 0.25. Because of addition of extra surface area needed for the heat transfer and the formation of these vortices, an enhancement in the local wall heat flux is observed especially in the downstream of each dimple. This effect causes the colder fluid move from central parts to the hotter zones of domain closer to the duct, thus enhancing the overall convective heat transfer.

REFERENCES