CHAPTER - 7

CFD VALIDATION OF EXPERIMENTATION - II
7.1 INTRODUCTION

The heat transfer from outer roller of Offset Printing machine has been the subject of numerous experimental and numerical investigations. In the present section, the effect of chilled water through the annular portion of roller on the outside temperature is studied. The three dimensional governing equations representing fluid flow and heat transfer are solved using finite volume based computational fluid dynamics (CFD) solver, FLUENT 6.3.26.

7.2 NUMERICAL MODELING

Navier –Stokes equations represent the transport of any variable in generic form. They are applicable to all types of flows, all modes of heat transfer, transport of species, with constant or variable properties. These equations appear in second order, non-linear, coupled partial differential form and hence, their analytical solution is difficult [21]. Computational Fluid Dynamics (CFD) codes solve these equations using numerical techniques. CFD is a powerful technique for the analysis of systems involving fluid flow, heat transfer and associated phenomena by means of computer based simulation. CFD codes are structured around the numerical algorithms and contain following three main elements.

- Pre-processor
- Solver
- Post-processor

The finite volume numerical formulation consists of following key steps:

1. Formal integration of the governing equations of fluid flow over all the (finite) control volumes of the solution domain and conservation of relevant properties for each finite size cell.

2. Discretization over the solution points which involves the substitution of a variety of finite difference type approximations for the gradient terms in the integrated equation representing flow processes such as convection, diffusion and sources. This converts the integral equations into a system of algebraic equations.

3. Solution of the algebraic equations by an iterative method.
CFD codes contain discretization techniques suitable for convection (transport due to fluid flow), diffusion (transport due to variation of $\Phi$ from point to point), source term (associated with the creation or destruction of $\Phi$) and rate of change with time. The underlying physical phenomenon is complex, non-linear and multidimensional. Hence, an iterative approach is adopted. The post-processing represents data extraction for visualization and reporting purpose. The facilities offered are:

- Domain geometry and grid display
- Vector plots
- Line and shaded contour plots
- Pathlines

7.3 GOVERNING EQUATIONS

The forced convection heat transfer under investigation is modeled by a set of partial differential equations describing the conservation of mass, momentum and energy in three-dimensional Cartesian coordinate system [21].

The set of equations allows specification of properties of ink and water viz. thermal conductivity, viscosity and specific heat. To account the turbulence effect, fluctuating component is added to the average velocity component. These modified equations are called as Reynolds Averaged Navier Stokes (RANS) equations. These are mentioned below:

$x$-momentum: \[
\frac{\partial (\rho \mathbf{U})}{\partial t} + \text{div}(\rho \mathbf{U} \mathbf{U}) = -\frac{\partial P}{\partial x} + \text{div}(\mu \text{ grad } \mathbf{U}) + S_m,
\]

\[+ \left[ -\frac{\partial (\rho u'^2)}{\partial x} - \frac{\partial (\rho u'v')}{\partial y} - \frac{\partial (\rho u'w')}{\partial z} \right] \]

$y$-momentum: \[
\frac{\partial (\rho \mathbf{V})}{\partial t} + \text{div}(\rho \mathbf{V} \mathbf{U}) = -\frac{\partial P}{\partial y} + \text{div}(\mu \text{ grad } \mathbf{V}) + S_m,
\]

\[+ \left[ -\frac{\partial (\rho v'^2)}{\partial x} - \frac{\partial (\rho v'w')}{\partial y} - \frac{\partial (\rho w'^2)}{\partial z} \right] \]

$z$-momentum: \[
\frac{\partial (\rho \mathbf{W})}{\partial t} + \text{div}(\rho \mathbf{W} \mathbf{U}) = -\frac{\partial P}{\partial z} + \text{div}(\mu \text{ grad } \mathbf{W}) + S_m,
\]

\[+ \left[ -\frac{\partial (\rho w'^2)}{\partial x} - \frac{\partial (\rho v'w')}{\partial y} - \frac{\partial (\rho w'^2)}{\partial z} \right] \]

Energy Equation: \[
\frac{\partial (\rho h)}{\partial t} + \nabla \cdot (\rho \mathbf{V} h) = \nabla \cdot (k_{eff} \nabla T) + S_h
\]
Where \( \rho \) is the density of the fluid, \( p \) is the pressure and \( \nu \) is the kinematic viscosity, \( k \) is the effective conductivity, \( h \) is the enthalpy, \( S_n \) represents the volumetric heat source.

These equations contain the additional stresses known as Reynolds stresses or turbulent stresses. These stresses can be estimated by solving the two equations for turbulent kinetic energy (\( k \)) and dissipation rate of turbulent kinetic energy (\( \varepsilon \)).

The numerical model is based on a control volume formulation. The above equations are integrated over each control volume to obtain a set of discretized linear algebraic equations, which are of the form:

\[
a_{ij}\phi_p = \sum a_{ij} \phi_{nb} + b
\]

The above set of algebraic equations describe processes affecting the value of \( \phi \) in cell \( P \) in relation to its neighbor cells together with the source term \( b \). These equations are solved by the commercially available CFD solver, FLUENT, employing SIMPLE algorithm for the pressure correction. Second order up wind schemes are used for interpolating velocity and temperature fields.

### 7.4 GEOMETRY CREATION AND MESH GENERATION

The setup consists of roller with ink layer on one side and chilled water flowing on the other side to cool the roller outer surface. The geometries with and without wire are meshed. The length of the roller is kept 1/5th of the actual length (the length containing only the one complete turn of wire) for the numerical modeling to save the computational time.

**Fig. 7.1 Various boundary conditions**

![Diagram showing various boundary conditions](image)

Fig. 7.1 shows the schematic configurations of modified roller with wire.

The set up has two continuums. The solid continuum refers to steel roller surrounding with copper coating. The fluid continuum consists of water flowing through the annular space between
wire and steel roller. The ink portion at the outermost region is also defined. Fig. 7.2 shows Grid on wire insert, inlet and outlet boundaries for the numerical study.

The Fluid and solid domain is meshed using 50000 tetrahedral cells to resolve the physics properly. These tetrahedral cells are then converted to polyhedral cells in FLUENT, so that the cell count is reduced and so the solver time. The grid quality is checked by the following parameters:

- Skewness
- Grid Area/Volume
- Aspect ratio
- Adjacent cell ratio

Fig. 7.2 Grid on wire insert, inlet and outlet boundaries

Fig. 7.3 Enlarged view of local section in fig No.7.2
The skewness is kept as low as possible (Maximum limit - 0.9) throughout the grid. The positive value of the area (2D) and volume (3D) is ensured to avoid overlapping of cells. The cell aspect ratio (ratio of longest edge length to shortest edge length) is kept as low as possible (Maximum limit -10). The adjacent cell ratio is controlled to ensure gradual growth of the adjacent cell size. The meshing of different regions of the domain and surfaces are shown in Fig.7.1 These meshing schemes are applicable to and adopted for all configurations. The mesh sizes used are as small as possible, consistent with the number of memory locations and the computational time available.

7.5 BOUNDARY CONDITIONS

All boundary conditions are implemented by the inclusion of additional source and/or sink terms at the boundaries during finite volume formulation. The imposed boundary conditions for all the configurations are as follows-

The volume of ink coating is given a uniform heat source 1469678 W/m$^3$. The water inlet is specified as a mass flow inlet with 0.0133 kg/s and temperature of the water at the inlet is given as 283 K. The outlet is assumed to be open to atmosphere (0 gauge pressure). The experimental convective heat input for the total length is 110.19 W. Therefore in numerical analysis the heat input is considered 25 W for a length of 130mm out of total length of 650mm.

**Calculation of source values**

Heat input given to test surface per unit volume= $Q/m^3= W / V$

\[
W/V = \frac{22}{\pi/4} \left(75^2 - 74^2\right) \times \left(10^{-3}\right)^2 \times 0.130 = 1469678 \text{ W/m}^3
\]
Material properties:

\[ k_{\text{copper}} = 387.6 \text{ W/mK} , \quad k_{\text{offset ink}} = 0.0167 \text{ W/mK} , \quad \mu = 0.08 \text{ kg/m-s} , \quad \rho = 849 \text{ kg/m}^3 \]

### 7.6 SOLVER

For segregated 3D second order steady solver, the SIMPLE Pressure-Velocity Coupling algorithms for the pressure correction process are used respectively. The discretization scheme used is standard, discretization for Pressure and second order upwind discretization for Momentum and Energy.

Ability to converge the results of numerical calculation means the imbalances in the iterative method have successfully fallen below the specified tolerance limits. The convergence can be associated a way, in which the solution not changing with the iterations. There are five residuals to be monitored in forced convection problem: continuity, X-velocity, Y-velocity, Z-velocity and energy. The default convergence criteria are 0.001 for all four of the above i.e. continuity and velocities and $10^{-6}$ for energy. Same values are used for first order scheme. Once the solution is converged in first order, during second order the convergence criteria are shifted to $10^{-6}$, as shown in Fig. 7.4. It is confirmed that beyond this limit, the changes in the average ink surface temperature is negligible.

![Residual convergence](image-url)
7.7 RESULTS OF CFD ANALYSIS

After obtaining the converged solutions, post processing was carried out. In this chapter various images of line and contour data is presented i.e. vector plots for velocity, temperature, contours of temperature, wall fluxes, surface heat transfer coefficient, velocity magnitude, density, path lines. One sample image of each scaled residuals and convergence history is also presented.

7.7.1 Contours of static temperature

Fig. 7.5 and 7.6 shows contours of static temperature for the surface temperature of ink without and with wire insert respectively. It is observed that the temperature gradient is visible at outmost annulus of water flow. From Fig. 7.5, it is observed that the temperature variation in bands across the length showing the non-uniformity and higher temperature variation over the length of the roller. The temperature variation of the ink surface affects the print quality, hence the helical wire insert which acts as turbulator is used in the annular space with the wire diameter of 3 mm. having the pitch of 130 mm. gives uniformity in ink surface temperature. Also it is observed that wire insertion gives closer temperature variation over length of the roller.

Fig. 7.5 Contours of static temperature, without wire
Fig. 7.6 Contours of static temperature, with wire

7.7.2 Contours of surface heat transfer coefficient

Fig. 7.7 shows contours of surface heat transfer coefficient for modified roller with wire insert; it is observed that most of the surface of the roller having the value of 75 W/m²K, which confirms the experimental results.

Fig. 7.7 Contours of surface heat transfer coefficient, with wire
7.7.3. Velocity vectors

Fig. 7.8 to fig 7.10 shows velocity vectors colored by ‘X’, ‘Y’ and ‘Z’ velocity (m/s). The range of magnitude of velocity is from minimum of 0.00046 m/sec to maximum of 1.58 m/sec. Rotational effect in annular space flow velocities is found to be dominating over the axial flow.
Fig. 7.10 Velocity vectors colored by ‘Z’ velocity magnitude (m/s)

7.4 Path lines

Fig. 7.11 to 7.13 shows path lines on the outer surface of water particles are released from inlet side of domain. Pathlines shows the turbulence due to insertion of wire, which enhance heat transfer coefficient. It shows the uniformity in the ink surface temperature as shown in Fig.7.6.

Fig. 7.11 Path lines on water outer, colored by particles
7.8 CLOSURE

In this chapter results obtained from CFD analysis are given. Results are presented in the form of various heat transfer parameters. It is concluded that, the values of average heat transfer coefficient $h_a$ for the given mass flow rate value of 0.8 lpm is around average value of 75 W/m²K. In modified experimentation value of $h_a$ is coming 69.50 W/m²K which closely matches with the computed value within 10%. The variation of temperature for the ink surface with wire insert is less by 3K than without wire insert. From various vector plots it is concluded that due to wire insertion heat transfer enhancement is achieved which results in uniformity in ink surface temperature.