To develop a Predictive Tool for Boiler Tube Failure

5. Cold Air Velocity Test

5.1 Need of Flue Gas Velocity and Temperature Prediction

Severe erosion of economizer tubes has been experienced in a number of utility boilers. The erosion typically occurs at the economizer tube bends and narrow tube passages of superheater platens. Fly Ash Erosion is one of the important causes of BTF. Particle velocity is the most important parameter as the rate of erosive loss is proportional to the velocity raised to an exponent that ranges between two and four. Particle velocity is driven by the local flow velocity at any particular boiler location. Sharpness factor of the eroding material plays important role in erosion failure and it depends on the temperature of the flue gases. Wear depends on the surface temperature. There are many parameters that can influence the tube surface temperature, including steam temperature and flow rate, deposit thickness and characteristics, and the rates of convective and radiation heat transfer from the flue gas and furnace cavity. These parameters are, in turn, strongly affected by the design and boiler operation. As a remedy to this failure many times solid shields and baffles are arbitrarily positioned in ‘areas’ where FAE occurred. It was recognized that the use of these palliative repair techniques was the main cause of repeat failures due to fly ash erosion. They simply redirected the high velocity flow onto an adjacent tube area. Abruptly placing the screens in the flue gases passage is not the solution. Hence, it is clear that to predict the fly ash erosion; it is required to know the velocity and temperature of flue gases in various zones of the boiler. The optimum long term solutions are based on identifying and reducing the highest velocity locations. It is not practical to measure the real time velocity and temperature of flue gases in all zones of the boiler when it is in operation due various operational difficulties. Size of boiler is too huge; the entire pipe length is more than 90 km, hundreds of tube coils; some of the places are so remote to access that an operator cannot physically reach the spot. If boiler tube failure takes place, then sometimes it is required to cut the un-failed tube coils to reach to the failed coil and repair it. Thus during repairs or the end of repair, unintentionally a new spot prone for failure is generated. Most of the time leakage is known after leakage takes place. There is no methodology in existence, which will help the engineers to predict the failure of an installed tube, predict the remnant life of the installed tube. If remnant life of tube is known, then during any forced outage, the
To develop a Predictive Tool for Boiler Tube Failure

tubes whose remnant life is on the verge of exhaust, can be attended for repairs of removal with new coils. Thus next unexpected forced outage is avoided and hence losses too are avoided.

This led to think of predictive tool for prediction of velocity of flue gases. Computational Fluid Dynamics was the obvious choice as the analysis involves fluid flow. However, owing to complex 3D geometry of the boiler, it was beyond the scope of the project to develop indigenous code for solving fluid flow equations. To predict the fly ash erosion, it is required to know the velocity and temperature of flue gases in various zones of the boiler. Hence, it was decided to use one of the commercially available CFD codes like FLUENT, CFX, etc. FLUENT was chosen for the purpose.

5.1.1 Modeling process [4]

A numerical model is provided with input data and is used to fix specific operating parameters and get the results. This seems to be very simple, and without further understanding, this simplistic view of modeling can lead to unsatisfactory results. It is systematic multistep process as explained below:

a. A complete situational report including physical geometry, process flow, physical property data and the level of solution is obtained. This detailed information is important because apparently small differences can make a significant effect on numerical solutions.

b. Appropriate modeling assumptions are made for the specific flow system and computer model selected and cost and time are balanced against level of details and information.

c. The general technical information obtained in step 1 is converted as input data into required by the computational model selected. Much of this is accomplished with the use of various computer programs such as computer assisted drafting (CAD) software and mesh generation software. Verification of the input data is an important part of this process.

d. The numerical computational model is run until an acceptable solution is obtained.

e. Results are analyzed to verify the initial model assumptions, to check the results against known trends, to benchmark the output with known field data, and to present the results in a usable form.

The complete CFD modeling is divided into following three steps:

(A) Pre-Processing
i. Solid Modeling
ii. Meshing the model.

(B) Solver
i. Set of governing equations (continuity, three momentum, energy and equation of state) to be used as per the application are selected.
ii. Physical models (e.g. turbulence, combustion models) are selected as per the application
iii. Material properties, boundary and initial conditions are set.

(C) Post-Processing
i. Temperature, velocity, pressure, etc profiles are plotted.
ii. Analysis of the plots and discussions on plots

Accordingly a complete geometrical model of entire 210 MW Pulverized Coal Fired Boiler is developed using ProE. Correctness of simulated results will depend on the correctness of the geometrical modeling. Hence it was necessary to get this geometrical model checked for its correctness and its suitability for CFD. It was known from the power plant authorities, that a Cold Air Velocity Test is conducted before commissioning of the boiler after the forced outage. Before commissioning of the boiler, forced draught and induced draught fans are run at 80% capacity. Velocity of the air (Cold Air) is physically measured at various points across various planes. These readings are tabulated and contours across the test planes are plotted. Abrupt changes in the velocity at sections or locations of increase in velocity across the pressure parts are known from the results. Corrective measures are thought off, suggested and incorporated. CAVT is again conducted after the repairs and modifications in flue gas path. If abrupt changes or increase in velocity is not noted then the boiler is commissioned. On the similar grounds the simulated CAVT is proposed to be performed. The results of experimental CAVT and simulated CAVT were compared and were used for verification of geometry, meshing and suggested procedure.

5.2 Cold Air Velocity Test (CAVT):

5.2.1 Pre requirements of CAVT
a) Past failure data, samples of failed tubes, test reports if any, and other problems known to contribute to erosion conditions – specific to areas of concern.
b) Original arrangement drawings of unit pressure part, (tube size, number and spacing), and drawings of modifications to convective pass identified (from original design), past, changed and present erosion control strategies.

c) Unit operating conditions: total gas flow at MCR, temperature profile at horizontal bank inlets.

d) A coal ultimate analysis (from representative sample) and fly ash analysis if available.

e) The secondary air fans (FD and ID) should be set (virtually) to at least 80 percent of MCR flow with normal damper settings for the CAVT.

f) Keep primary or other air sources off.

This operational data is in Annexure II

5.2.2 CAVT Procedure

a) Identify the test locations and test planes.

b) Locate these test planes on the drawing.

c) Set the inlet and outlet boundary conditions and virtually run FD and ID fans at

d) At each measurement plane, measurement lines and points are identified on drawings or in the unit as required. To get a representative sample, usually 70-100 points are selected for each plane.

e) Measurements are made between the first row of tubes at the inlet or outlet of each bank, and at the side, front and rear wall gaps.

f) At the end of the CAVT, data is examined and additional measurements are taken, if necessary, to verify initial measurements or to detail areas of concern.

5.2.3 Data Analysis/Design of Flow Controls

Data from the CAVT is converted to tables giving velocity measurements in m/s at test conditions, normalized to the plane average, and at the equivalent MCR gas velocities in m/s (all data will be reported in SI units). This equivalent gas velocity data is compared to the known erosion areas. Based upon this analysis and previous experience in flow control and erosion prevention, screens and/or baffles are selected to reduce peak levels of tube erosion in areas with the worst problems. General arrangement (GA) drawings showing the location, size and orientation of the flow
control devices would be prepared, along with appropriate materials lists. These
drawings will be suitable for Plant to develop detailed installation drawings.

**Test Reports**

A preliminary test report is issued following the CAVT and initial data
analysis to provide preliminary recommendations for erosion controls. This provides
utility personnel an opportunity to review initial results and to comment. A final
report is issued that includes the test results/analyses, recommended flow controls
with GA drawings, materials lists, installation notes and photographs from the unit
inspection and test.

Results of sample CAVT provide by plant authorities is provided in Annexure
III, which is used for comparison of the simulated CAVT results.

### 5.3 Cold Air Velocity Test (Simulated)

#### 5.3.1 Physical and Computational Domain:

Boiler under study is 210MW pulverized coal fired boiler. The Plant features a
Tangentially Fired, Balanced Draught, Natural Circulation and Direct Fired
Pulverized Coal fired Boiler with Blow Mills for each unit. Plant has Full Load
generating capacity 700 TPH of steam at 540°C and 137kg/cm² before inlet to high
pressure turbine. For each stage there is a set of high, intermediate and low pressure
turbines. HP turbine is single flow. Its first stage is impulse type and remaining stages
are reaction type. It has 12 stages. IP turbine is single flow reaction type. It has
11stages.LP turbine is double flow reaction type with 4 stages in each flow. The plant
employs regenerative feed heating system along with environmental pollution control
devices such as electrostatic precipitator to utilize heat of expanded steam and flue
gases before air pre-heater. Flue gases then lead to chimney. Face at entry of WWP
was considered as inlet face and, face before Draught Fan was considered as outlet
face of computational domain. The plant under study is discussed in detail and depth
in chapter 3.

Geometric modelling is the very first step in any CFD analysis. First of all the
boiler drawing provided by the plany authorities was studied in detail. The cross-
section on furnace side is 13868mm(depth)×10592mm(width) and vertical dimension
is 19417mm. Total width is 24230mm. Vertical height on draught fan side is
18507mm and cross-section is 8255mm×13868mm. The space occupied by gas after
combustion chamber and before air pre-heater was treated as control volume. Since it
To develop a Predictive Tool for Boiler Tube Failure

was decided to study fluid flow through this medium it is treated to be solid and meshed for further analysis. Since the model is mostly solid Pro-E Wildfire 3.0 is used as modelling software. All the dimensions of model are in mm. Model was created as a Single Part and the flue gas space is of concern, space occupied by Boiler Tubes (dia equal to outer dia) was removed from model i.e. control volume.

First component in flue gas path is Water wall Platen Super-Heater. Extreme planes are at a distance of 1562 mm from wall on both sides. There are 41 tubes in one set of WWP tubes separated by distance of 73.5mm. There are 4 sets of WWP with pitch of 3581 mm. Outside diameter of tubes is 63.5 mm. Total length of tubes is 560306 mm. Second component in flue gas path is Platen Super-Heater. Extreme planes are at a distance of 534 mm from wall on both sides. One set of PSH consists of 24 tubes separated by distance of 60.5mm. There are 29 sets of PSH with pitch of 457 mm. Outside diameter of tubes is 51 mm. Total length of tubes is 5945632 mm.

Third component in flue gas path is Cold Re-Heater. Extreme plane is at a distance of 248 mm from wall on both sides. One set of CRH consists of tubes separated by distance of 60.5 mm. There are total 59 sets with pitch of 23.5 mm. Outside diameter of tubes is 54mm. Total length of tubes is 139873.8mm.

Fourth component in flue gas path is Hot Re-Heater. Extreme plane is at a distance of 248 mm from wall on both sides. One set of HRH consists of 16 tubes separated by distance of 114.5 mm. There are total 89 sets with pitch of 152 mm. Outside diameter of tubes is 54 mm. Total length of tubes is 135642.3mm.

Fig.5.1 (a) 2D view of the boiler model
Fifth component in flue gas path is Final Super-Heater. Extreme plane is at a distance of 178.5 mm from wall on both sides. One set of HRH consists of 8 tubes separated by distance of 114.5 mm. There are total 119 sets with pitch of 114.5 mm. Outside diameter of tubes is 47.63 mm. Total length of tubes is 56760 mm.

Sixth component in flue gas path is Low Temperature Super-Heater. Extreme plane is at a distance of 184 mm from wall on both sides. One set of HRH consists of 16 tubes separated by distance of 114.5 mm. There are total 89 sets with pitch of mm. Outside diameter of tubes is 54 mm. Total length of tubes is 135642 mm.

Seventh component in flue gas path is Economizer. It is Continuous Finned Tube Staggered Arrangement, single stage, with 270 tube coils and with 5690 m². The Tube diameter & thickness is 44.5 mm OD x 4.5 mm thick and made of SA 210 Gr. A1.

Geometrical drawings of Boiler and pressure parts are attached as Annexure IV. The Pro-E model of boiler is further exported to the meshing software.

5.3.2 Meshing:
Meshing is the intermediate step between the geometric modeling and the flow analysis. Using Gambit 2.4.6 geometry is meshed. First “.iges” model of Pro-E is imported in gambit with scaling factor of 0.001 to convert geometry from mm to m.
Due to constraints of fine geometry near tubes and dense arrangement of tubes, the cell count required was very high (more than 50 million) and hence the entire geometry could not be meshed. The computational requirements to meet these conditions like RAM, processor speed, etc. were not available. In order to overcome this situation, it was decided to simplify the model. Literature survey revealed the porous media concept which could be used to solve such tube banks problems. In this approach, the tube bank is replaced with a porous block having equivalent overall dimensions and properties like porosity, inertial resistance, etc. which are calculated from original tube bank model. This approach is discussed in detail in subsequent chapter.

Since in this small gaps in tubes were eliminated, meshing was possible. After importing and scaling porous model in gambit, edges were created bounding the porous blocks. Edges were joined to form surfaces which in turn formed respective volumes porous parts of boiler. While creating porous volumes, the outer boundaries of the tube were used. In case of CRH and HRH, there was gap between the two. As the pressure drops across the CRH and HRH were not known individually, the inertial resistance was calculated for RH as a whole. However, while analyzing the model in FLUENT, it was observed that the inertial resistance applied to the RH was being encountered twice by the flow of flue gases due to the gap between CRH and HRH. This caused additional pressure drop which was undesired. Hence, it was decided to combine CRH and HRH into single component.

**Meshing Procedure**

Various combinations of edge, surface and volume meshing were tried and following method was found to be best. Also by this method cell count was not too large for operating system. All edges were selected and interval size of 300 mm with successive ratio of 1 meshing was done. For surface meshing all surface were selected, meshing scheme was with ‘Tri’ elements and ‘Pave’ type. Default spacing was used so as it takes previous element size. Finally for volume meshing, scheme was ‘Tetra/hybrid’ elements and ‘TGrid’ type with default spacing. Face at entry of WWP was given ‘Pressure Inlet’ as boundary condition, face before Draught Fan was given condition as ‘Pressure Outlet’ while remaining external surfaces was given ‘Wall’ condition. The mesh was exported as “.msh” file which can be read in the FLUENT for subsequent analysis and post-processing. The quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. The
attributes associated with mesh quality are node point distribution, smoothness, and skewness. After completing the mesh its quality was checked.

Computational facilities and computational time are the main constraints in the simulation of such huge models. Hence porous media approach is used for modeling and it is then meshed.

5.4 Porous Media Approach [45, 46]:

Porous media can be used for modeling a wide variety of engineering applications, including flows through packed beds, filters, perforated plates, flow distributors, and tube banks.

![Fig. 5.2 Meshed Geometry](image)

It is generally desirable to determine the pressure drop across the porous medium and to predict the flow field in order to optimize a given design. In this case, steady 3D flow through porous media is modeled. The dimensions of the non porous and porous zone are specified. The flow can be modeled as laminar or turbulent depending on the Reynolds number. Medium mesh option is used due inadequate computational facilities. The material properties for the fluid and porous medium (porosity, viscous and inertial resistance) are specified.

5.4.1 Basic Principles:

In the region of a porous medium, the Navier-Stokes-equation extends to

\[
\rho \left[ \frac{1}{\phi} \frac{\partial}{\partial t} \mathbf{v} + \frac{1}{\phi^2} (\mathbf{v} \nabla) \mathbf{v} \right] = -\nabla p + \eta_e \nabla^2 \mathbf{v} + \rho g \left( \frac{\eta}{K} + a \rho |\mathbf{v}| \right) \nabla \mathbf{v}
\]

............. (27)
To develop a Predictive Tool for Boiler Tube Failure

Which has the additional parameters porosity, \( \varphi \), effective viscosity, \( \eta_e \), Forchheimer-coefficient, \( a \), and permeability, \( [K] \) have to be taken into account. The porosity \( \varphi \) is defined as

\[
\varphi = \frac{V_{\text{free}}}{V_{\text{total}}} \tag{28}
\]

It describes the fraction of the free remaining volume with respect to the total volume. It is also defined as the ratio of void volume in porous medium sample to total volume of porous medium sample.

Properties such as porosity, viscous resistance and inertial resistance for any porous media of interest are required to be calculated. The mean inlet velocity was maintained constant and the porous media properties were varied to obtain the desired pressure drop per unit length.

The engineering heat transfer results assembled in this chapter refer primarily to fluid saturated porous media that can be modeled as non-deformable, homogeneous, and isotropic. In such media, the volumetric porosity \( \varphi \) is the same as the area ratio (void area contained in the sample cross section)/ (total area of the sample cross section). The phenomenon of convection through the porous medium is described in terms of volume-averaged quantities such as temperature, pressure, concentration, and velocity components. Each volume-averaged quantity \( (\psi) \) is defined through the operation

\[
\langle \psi \rangle = \frac{1}{V} \iiint \psi \, dV \tag{29}
\]

where \( \psi \) is the actual value of the quantity at a point inside the sample volume \( V \). Alternatively, the volume-averaged quantity equals the value of that quantity averaged over the total volume occupied by the porous medium. The volume sample is called representative elementary volume (REV).

5.4.2 Mass Conservation

The principle of mass conservation or mass continuity applied locally in a small region of the fluid-saturated porous medium is

\[
\frac{D\rho}{Dt} + \rho \nabla \cdot \mathbf{v} = 0 \tag{30}
\]
Where, $D/Dt$ is the material derivative operator.

\[
\frac{D}{Dt} = \frac{\partial}{\partial t} + u \frac{\partial}{\partial x} + v \frac{\partial}{\partial y} + w \frac{\partial}{\partial z}
\]

(31)

Where, $\mathbf{v} (u, v, w)$ is the volume-averaged velocity vector. For example, the volume-averaged velocity component $u$ in the $x$ direction is equal to $\phi u_p$, where $u_p$ is the average velocity through the pores. In many single-phase flows through porous media, the density variations are small enough so that the $\frac{D\rho}{Dt}$ term may be neglected in eq. (5.4).

### 5.4.3 Assumptions used and Limitations of the Porous Media Model

The porous media model can be used for a wide variety of problems, including flows through packed beds, filter papers, perforated plates, flow distributors, and tube banks. In this case, a tube bank is replaced with a porous block having equivalent overall dimensions. (Fig 5.3) For using this model, a cell zone is defined in which the porous media model is applied and the pressure loss in the flow is determined via user inputs. Heat transfer through the medium can also be represented, subject to the assumption of thermal equilibrium between the medium and the fluid flow.

The porous media model incorporates an empirically determined flow resistance in a region of model defined as "porous". The porous media model is an added momentum sink in the governing momentum equations. As such, the following modeling assumptions and limitations are recognized:

Fig. 5.3 Geometrical Model of Boiler with Porous Block
To develop a Predictive Tool for Boiler Tube Failure

i) Since the volume blockage that is physically present is not represented in the model, by default FLUENT uses and reports a superficial velocity inside the porous medium, based on the volumetric flow rate, to ensure continuity of the velocity vectors across the porous medium interface. As a more accurate alternative, the true (physical) velocity is used inside the porous medium.

ii) The effect of the porous medium on the turbulence field is only approximated.

iii) While performing the heat transfer analysis, it will be required to add a source term. The relative reference frame rather than the absolute reference frame is used to obtain the correct source terms.

iv) When specifying the specific heat capacity, CP, for air in the porous zone, CP is entered as a constant value.

5.4.4 Continuity and Momentum Equation

Governing Equations

Mass conservation statement:

Rate of increase of mass in fluid element = Net rate of flow of mass into fluid element

Fig. 5.4 Section Model of Model of Boiler with Pressure Parts
To develop a Predictive Tool for Boiler Tube Failure

Governing equation for above statement is

$$\frac{\partial \rho}{\partial t} + \left( \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} \right) = 0 \quad \text{............................................................ (32)}$$

This is the unsteady, three-dimensional mass conservation or continuity equation at a point in a compressible fluid. The first term on the left-hand side is the rate of change of the density. The second term indicates the net flow of mass out of the element across its boundaries and is called the convective term.

Momentum conservation statement:

Rate of increase of momentum of fluid particle = Sum of forces on fluid particle

X-momentum

$$\rho \frac{Du}{Dt} = \frac{\partial (-p + \tau_{xx})}{\partial x} + \frac{\partial (\tau_{xy})}{\partial y} + \frac{\partial (\tau_{xz})}{\partial z} + S_{Mx} \quad \text{............................................................ (33)}$$

Y-momentum

$$\rho \frac{Dv}{Dt} = \frac{\partial (\tau_{yx})}{\partial x} + \frac{\partial (-p + \tau_{yy})}{\partial y} + \frac{\partial (\tau_{yz})}{\partial z} + S_{My} \quad \text{............................................................ (34)}$$

Z-momentum

$$\rho \frac{Dw}{Dt} = \frac{\partial (\tau_{zx})}{\partial x} + \frac{\partial (\tau_{zy})}{\partial y} + \frac{\partial (-p + \tau_{zz})}{\partial z} + S_{Mz} \quad \text{............................................................ (35)}$$

The sign associated with the pressure shows compressive nature and that associated with the viscous stress shows tensile nature.

$S_{Mi}$ indicates source term due to body forces that is gravitational, electromagnetic, centrifugal etc.

Energy conservation statement:

$$\rho \frac{Di}{Dt} = -p \, div \mathbf{u} + div (k \, grad \, T) + \tau_{xx} \frac{\partial u}{\partial x} + \tau_{xy} \frac{\partial u}{\partial y} + \tau_{xz} \frac{\partial u}{\partial z} + \tau_{yx} \frac{\partial v}{\partial x} + \tau_{yy} \frac{\partial v}{\partial y} + \tau_{yz} \frac{\partial v}{\partial z} + \tau_{zx} \frac{\partial w}{\partial x} + \tau_{zy} \frac{\partial w}{\partial y} + \tau_{zz} \frac{\partial w}{\partial z} + S_i \quad \text{............................................................ (36)}$$

Here we have five equations in the terms of six unknown flow field variable $\rho$, $p$, $u$, $v$, $w$, $i$. Thus Equations of State are used to relate the other variables to the two state variables. If $\rho$ and $T$ are used as state variables the equation of state are
To develop a Predictive Tool for Boiler Tube Failure

This provides sixth equation but also introduce the seventh unknown temperature, T. A seventh equation to close the entire system must be a thermodynamic relation between state variable

\[ i = i (\rho, T) \]

These governing equations contain as further unknowns the viscous stress components, \( \tau_{ij} \). In fluid flows the viscous stresses can be expressed as functions of the local deformation rate (or strain rate). In three-dimensional flows the local rate of deformation is composed of the linear deformation rate and the volumetric deformation rate.

The rate of linear deformation of a fluid element has nine components in three dimensions denoted by \( e_{ij} \).

There are three elongating deformation components:

\[
e_{xx} = \frac{\partial u}{\partial x}, \quad e_{yy} = \frac{\partial v}{\partial y}, \quad e_{zz} = \frac{\partial w}{\partial z}
\]

...................................................................................... (37)

There are also six shearing linear deformation components:

\[
e_{xy} = e_{yx}, \quad e_{xz} = e_{zx}, \quad e_{yz} = e_{zy}
\]

\[
e_{xy} = e_{yx} = \frac{1}{2} \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right), \quad e_{xz} = e_{zx} = \frac{1}{2} \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right), \quad e_{yz} = e_{zy} = \frac{1}{2} \left( \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right)
\]

........ (38)

The volumetric deformation is given by

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = \text{div} \mathbf{u}
\]

...................................................................................... (39)

In a Newtonian fluid the viscous stresses are proportional to the rates of deformation. There are two constant of proportionality: the dynamic viscosity, \( \mu \), to relate stresses to linear deformations, and second viscosity, \( \lambda \), to relate stresses to the volumetric deformations. Thus:

\[
\begin{align*}
\tau_{xx} &= 2\mu \frac{\partial u}{\partial x} + \lambda \text{div} \mathbf{u} \\
\tau_{yy} &= 2\mu \frac{\partial v}{\partial y} + \lambda \text{div} \mathbf{u} \\
\tau_{zz} &= 2\mu \frac{\partial w}{\partial z} + \lambda \text{div} \mathbf{u}
\end{align*}
\]

......................................................................................(40)
To develop a Predictive Tool for Boiler Tube Failure

\[
\begin{align*}
\tau_{xy} &= \tau_{yx} = \mu \left( \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right), \\
\tau_{xz} &= \tau_{zx} = \mu \left( \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \right), \\
\tau_{yz} &= \tau_{zy} = \mu \left( \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right).
\end{align*}
\]

\[\text{...............................................................................(41)}\]

For gases a good working approximation is: \( \lambda = -2/3 \mu \)

Substitution of the above shear stresses into X-momentum equation yields

\[
\rho \frac{Du}{Dt} = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left[ 2\mu \frac{\partial u}{\partial x} + \lambda \text{div } \mathbf{u} \right] + \frac{\partial}{\partial y} \left[ \mu \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \right] + \frac{\partial}{\partial z} \left[ \mu \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right) \right] + S_{ux} \]

\[\text{...............................................................................(42)}\]

Rearranging the viscous stresses terms the equation becomes (in more compact form):

\[
\begin{align*}
\rho \frac{Du}{Dt} &= -\frac{\partial p}{\partial x} + \text{div} (\mu \text{ grad } u) + S_{ux}, \\
\rho \frac{Dv}{Dt} &= -\frac{\partial p}{\partial y} + \text{div} (\mu \text{ grad } v) + S_{vy}, \\
\rho \frac{Dw}{Dt} &= -\frac{\partial p}{\partial z} + \text{div} (\mu \text{ grad } w) + S_{wz}.
\end{align*}
\]

\[\text{...............................................................................(43)}\]

If the Newtonian model of viscous stresses is used in the internal energy equation, the equation, after some rearrangement, appears

\[
\rho \frac{Di}{Dt} = -p \text{div } \mathbf{u} + \text{div} (\mu \text{ grad } T) + \phi + S_i \]

\[\text{...............................................................................(44)}\]

where,

\[
\phi = \mu \left[ 2 \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial u}{\partial y} \right)^2 + \left( \frac{\partial u}{\partial z} \right)^2 + \left( \frac{\partial v}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial y} \right)^2 + \left( \frac{\partial v}{\partial z} \right)^2 + \left( \frac{\partial w}{\partial x} \right)^2 + \left( \frac{\partial w}{\partial y} \right)^2 + \left( \frac{\partial w}{\partial z} \right)^2 \right] + \lambda (\text{div } \mathbf{u})^2.
\]

\[\text{...............................................................................(45)}\]

It is a dissipation function and represents a source of internal energy due to deformation work on the fluid particle. This work is extracted from the mechanical agency, which causes the motion and is converted to internal energy or heat.

**Turbulence and its Modelling**

The flow encountered in boiler is turbulent. Turbulent flows are characterized by fluctuating velocity fields. These fluctuations mix transported quantities such as momentum, energy, and species concentration, and cause the transported quantities to fluctuate as well. Since these fluctuations can be of small scale and high frequency, they are too computationally expensive to simulate directly in practical engineering
To develop a Predictive Tool for Boiler Tube Failure

calculations. Instead, the instantaneous (exact) governing equations can be time-averaged, ensemble-averaged, or otherwise manipulated to remove the small scales, resulting in a modified set of equations that are computationally less expensive to solve. However, the modified equations contain additional unknown variables, and turbulence models are needed to determine these variables in terms of known quantities.

**Continuity equation**

\[
\frac{\partial \rho}{\partial t} + \text{div} (\rho U) = 0
\]

…………………………………………………………….. (46)

**Momentum equations**

\[
\begin{align*}
\frac{\partial (\rho u)}{\partial t} + \text{div} (\rho u U) & = -\frac{\partial \rho}{\partial x} + \text{div} (\mu \text{grad} u) + \left[ -\frac{\partial (\rho u' u)}{\partial x} - \frac{\partial (\rho v' v)}{\partial y} - \frac{\partial (\rho w' w)}{\partial z} \right] + S_{\text{ext}} \\
\frac{\partial (\rho v)}{\partial t} + \text{div} (\rho v U) & = -\frac{\partial \rho}{\partial y} + \text{div} (\mu \text{grad} v) + \left[ -\frac{\partial (\rho u' v')}{\partial x} - \frac{\partial (\rho v' v')}{\partial y} - \frac{\partial (\rho w' v')}{\partial z} \right] + S_{\text{m}_{y}} \\
\frac{\partial (\rho w)}{\partial t} + \text{div} (\rho w U) & = -\frac{\partial \rho}{\partial z} + \text{div} (\mu \text{grad} w) + \left[ -\frac{\partial (\rho u' w')}{\partial x} - \frac{\partial (\rho v' w')}{\partial y} - \frac{\partial (\rho w' w')}{\partial z} \right] + S_{\text{m}_{z}}
\end{align*}
\]

Which in general notation

\[
-\rho u_{i} u_{j} = \tau_{ij} = \mu_{T} \left( \frac{\partial u_{i}}{\partial x_{j}} + \frac{\partial u_{j}}{\partial x_{i}} \right)
\]

…………………………………………………………….. (48)

**Choosing a Turbulence Model**

No single turbulence model is universally accepted as being superior for all classes of problems. The choice of turbulence model will depend on considerations such as the physics encompassed in the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation. To make the most appropriate choice of model it is needed to understand the capabilities and limitations of the various options. The different turbulence models available are:

1) Spalart – Allmaras
2) Standard K-Epsilon
3) Realizable K-Epsilon
4) RNG K-Epsilon
5) Standard K-Omega

6) SST K- Omega.

Of the above models K-Epsilon are best suited for boiler flow problems. A brief description about K-Epsilon is provided below. A turbulence model is a computational procedure to close the system of mean flow and solve them, so that a more or less wide variety of flow problems can be calculated. There are some well known classical turbulence models such as the mixing length, k–ε family models, Reynolds stress and algebraic stress models. In the most of the CFD research the k–ε family models including; the standard, RNG and realizable model are used in the simulation. The flow in a utility boiler is turbulent and the Reynolds number based on the average velocity always exceeds the critical value. Hence it is recommended to use k–ε turbulence model. The standard k–ε turbulence model is used due to complicated mathematical work and need to employ a model which it uses lower CPU time. The model provides the transport equations for k and ε as follows:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \rho \frac{v_{eff}}{\sigma_k} \frac{\partial k}{\partial x_i} \right) + \rho (P_{k-\varepsilon}) \quad \ldots \ldots \ldots (49)
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho u_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \rho \frac{v_{eff}}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_i} \right) + S_\varepsilon \quad \ldots \ldots \ldots (50)
\]

\[
\begin{align*}
\mu_{eff} &= \mu + \mu_t \\
\mu_t &= \rho C_\mu \frac{k^2}{\varepsilon}
\end{align*}
\]

\[
\left\{ C_1, C_2, C_\mu \right\}
\]

In which

\[
\begin{align*}
S_\varepsilon &= \rho \left( C_1 \frac{\varepsilon}{k} P_k - C_2 \frac{\varepsilon^2}{k} \right) \\
P_k &= \nu_i \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \frac{\partial u_i}{\partial x_j} \right)
\end{align*}
\]

Where

C1, C2, C_\mu constants of the k–ε model

U velocity vector

u, v, w, u_i, u_j mean velocity components

u’, v’, w’, u’_i, u’_j turbulent fluctuating velocity component

x_i, x_j Cartesian coordinate
ε dissipation rate of k
ρ density
μ, μT, μeff laminar, turbulent and effective viscosities
σk, σε turbulent Prandtl numbers for k–ε

**Momentum Equations for Porous Media**

Porous media are modeled by the addition of a momentum source term to the standard fluid flow equations. The source term is composed of two parts: a viscous loss term and an inertial loss term.

\[
S_i = -\left(\sum_{j=1}^{3} D_{ij} \mu \partial_j + \sum_{j=1}^{3} C_{ij} \frac{1}{2} \rho \partial_{mag} \partial_j \right) \quad \text{.........} \quad (53)
\]

Where, \( S_i \) is the source term for the \( i \)th \((x, y \text{ or } z)\) momentum equation, and \( D \) and \( C \) is prescribed matrices. This momentum sink contributes to the pressure gradient in the porous cell, creating a pressure drop that is proportional to the fluid velocity (or velocity squared) in the cell.

To recover the case of simple homogeneous porous media

\[
S_i = -\left(\frac{\mu}{\alpha} \partial_i + C_2 \frac{1}{2} \rho \partial_{mag} \partial_i \right) \quad \text{.........} \quad (54)
\]

Where, \( \alpha \) is the permeability and \( C_2 \) is the inertial resistance factor, simply specify \( D \) and \( C \) as diagonal matrices with \( 1/\alpha \) and \( C_2 \), respectively, on the diagonals (and zero for the other elements).

FLUENT also allows the source term to be modeled as a power law of the velocity magnitude:

\[
S_i = -C_0 |v|^C_1 = -C_0 |v|^{[C_1-1]} v_i \quad \text{.........} \quad (55)
\]

Where, \( C_0 \) and \( C_1 \) are user-defined empirical coefficients. In the power-law model, the pressure drop is isotropic and the units for \( C_0 \) are SI.

**Darcy's Law in Porous Media**

In laminar flows through porous media, the pressure drop is typically proportional to velocity and the constant \( C_2 \) can be considered to be zero. Ignoring convective acceleration and diffusion, the porous media model then reduces to Darcy's Law:

\[
\nabla p = -\frac{\mu}{\alpha} \mathbf{\nabla} \quad \text{.........} \quad (56)
\]

The pressure drops that FLUENT computes in each of the three \((x, y, z)\) coordinate directions within the porous region is then
\[ \Delta p_x = \sum_{j=1}^{3} \frac{\mu}{\alpha_{xj}} v_j \Delta n_x \]  

(57)

\[ \Delta p_y = \sum_{j=1}^{3} \frac{\mu}{\alpha_{yj}} v_j \Delta n_y \]  

(58)

\[ \Delta p_z = \sum_{j=1}^{3} \frac{\mu}{\alpha_{zj}} v_j \Delta n_z \]  

(59)

where, \(1/\alpha_{xj}\) are the entries in the matrix \(D\) in Eq. 5.32 \(5.33\) and \(5.34\), \(v\) are the velocity components in the \(x\), \(y\), and \(z\) directions, and \(\Delta n_x\), \(\Delta n_y\), and \(\Delta n_z\) are the thicknesses of the medium in the \(x\), \(y\), and \(z\) directions. Here, the thickness of the medium \((\Delta n_x, \Delta n_y, \text{or} \Delta n_z)\) is the actual thickness of the porous region in the model.

### 5.4.5 Inertial Losses in Porous Media

At high flow velocities, the constant \(C_2\) in Eq. 5.29 provides a correction for inertial losses in the porous medium. This constant can be viewed as a loss coefficient per unit length along the flow direction, thereby allowing the pressure drop to be specified as a function of dynamic head.

While modeling a perforated plate or tube bank, we can sometimes eliminate the permeability term and use the inertial loss term alone, yielding the following simplified form of the porous media equation:

\[ \nabla p = -\sum_{j=1}^{3} C_{2ij} \left( \frac{1}{2} \rho v_j v_{mag} \right) \]  

(60)

Or when written in terms of the pressure drop in the \(x\), \(y\), and \(z\) directions:

\[
\begin{align*}
\Delta p_x & \approx \sum_{j=1}^{3} C_{2xj} \Delta n_x \frac{1}{2} \rho v_j v_{mag} \\
\Delta p_y & \approx \sum_{j=1}^{3} C_{2yj} \Delta n_y \frac{1}{2} \rho v_j v_{mag} \\
\Delta p_z & \approx \sum_{j=1}^{3} C_{2zj} \Delta n_z \frac{1}{2} \rho v_j v_{mag}
\end{align*}
\]  

(61)

Again, the thickness of the medium \((\Delta n_x, \Delta n_y, \text{or} \Delta n_z)\) is the thickness defined in the model.

### 5.4.6 Treatment of the Energy Equation in Porous Media

FLUENT solves the standard energy transport equation in porous media regions with modifications to the conduction flux and the transient terms only. In the porous medium, the conduction flux uses an effective conductivity and the transient term includes the thermal inertia of the solid region on the medium:
\[
\frac{\partial}{\partial t} \left( \gamma \rho_f E_f + (1 - \gamma) \rho_s E_s \right) + \nabla \cdot \left( \bar{\nabla} (\rho_f E_f + p) \right) \nabla \left[ k_{\text{eff}} \nabla T - \left( \sum_i h_i J_i \right) + (\bar{r} \cdot \bar{v}) \right] S^h_f. \tag{62}
\]

where

- \( E_f \) = total fluid energy
- \( E_s \) = total solid medium energy
- \( \gamma \) = porosity of the medium
- \( k_{\text{eff}} \) = effective thermal conductivity of the medium
- \( S^h_f \) = fluid enthalpy source term

### 5.4.7 Effective Conductivity in the Porous Medium

The effective thermal conductivity in the porous medium, \( k_{\text{eff}} \), is computed by FLUENT as the volume average of the fluid conductivity and the solid conductivity:

\[
k_{\text{eff}} = \gamma k_f + (1 - \gamma) k_s. \tag{63}
\]

Where,

- \( \gamma \) = porosity of the medium
- \( k_f \) = fluid phase thermal conductivity (including the turbulent contribution, \( k_t \))
- \( k_s \) = solid medium thermal conductivity

#### Treatment of Turbulence in Porous Media

FLUENT, by default, solves the standard conservation equations for turbulence quantities in the porous medium. In this default approach, therefore, turbulence in the medium is treated as though the solid medium has no effect on the turbulence generation or dissipation rates. This assumption may be reasonable if the medium's permeability is quite large and the geometric scale of the medium does not interact with the scale of the turbulent eddies.

### Effect of Porosity on Transient Scalar Equations

For transient porous media calculations, the effect of porosity on the time-derivative terms is accounted for in all scalar transport equations and the continuity equation. When the effect of porosity is taken into account, the time-derivative term becomes \( \frac{\partial}{\partial t} (\gamma \rho \phi) \), where \( \phi \) is the scalar quantity (\( k, \varepsilon, \) etc.) and \( \gamma \) is the porosity. The effect of porosity is enabled automatically for transient calculations.
User Inputs for Porous Media

While modeling a porous region, the only additional inputs for the problem setup are as follows. Optional inputs are indicated as such.

a. The porous zone is defined.

b. The porous velocity formulation is defined.

c. The fluid material flowing through the porous medium is identified.

d. Enabling reactions for the porous zone, and the reaction mechanism is selected.

e. Set the viscous resistance coefficients and the inertial resistance coefficients and define the direction vectors for which they apply. Alternatively, specify the coefficients for the power-law model.

f. The porosity of the porous medium is specified.

g. The material contained in the porous medium is selected (required only for models that include heat transfer). Note that the specific heat capacity, $C_p$, for the selected material in the porous zone can only be entered as a constant value.

h. The volumetric heat generation rate in the solid portion of the porous medium (or any other sources, such as mass or momentum). (Optional)

i. Any fixed values for solution variables in the fluid region are set. (Optional)

Methods for determining the resistance coefficients and/or permeability are presented below. If we choose to use the power-law approximation of the porous-media momentum source term, we will enter the coefficients $C_0$ and $C_1$ instead of the resistance coefficients and flow direction.

5.5 Modeling Porous Media Based On Physical Velocity

FLUENT calculates the superficial velocity based on volumetric flow rate. The superficial velocity in the governing equations can be represented as

$$
\bar{v}_{\text{superficial}} = \gamma \bar{v}_{\text{physical}}
$$

Where, $\gamma$ is the porosity of the media defined as the ratio of the volume occupied by the fluid to the total volume.

The superficial velocity values within the porous region remain the same as those outside of the porous region. This limits the accuracy of the porous model where there should be an increase in velocity throughout the porous region. For more accurate simulations of porous media flows, it becomes necessary to solve for the
true, or physical velocity throughout the flow-field, rather than the superficial velocity.

Using the physical velocity formulation, and assuming a general scalar $\phi$, the governing equation in an isotropic porous media has the following form:

$$ \frac{\partial (\gamma \rho \psi \phi)}{\partial t} + \nabla \cdot (\gamma \rho \psi \phi) = \nabla \cdot (\gamma \nabla \phi) + \gamma S_{\phi} \hspace{1cm} (64) $$

Assuming isotropic porosity and single phase flow, the volume-averaged mass and momentum conservation equations are as follows:

$$ \frac{\partial (\gamma \rho \psi)}{\partial t} + \nabla \cdot (\gamma \rho \psi \tilde{\psi}) = \hspace{1cm} (65) $$

$$ \frac{\partial (\gamma \rho \tilde{\psi})}{\partial t} + \nabla \cdot (\gamma \rho \tilde{\psi} \tilde{\psi}) = -\gamma \nabla p + \nabla \cdot (\gamma \tilde{\psi} \tilde{\psi}) + \gamma \tilde{B}_f - \left( \frac{\mu}{\alpha} + \frac{C_2 \rho}{2} |\tilde{\psi}| \right) \tilde{\psi} \hspace{1cm} (66) $$

The last term in 5.41 represents the viscous and inertial drag forces imposed by the pore walls on the fluid. [47]

**Porous Block Modeling**

Fig. 5.3 shows the 3D cut section of the boiler with blocks removed from it which will later be treated as porous medium. In this case, PSH, RH (CRH & HRH combined), FSH, LTSH and ECONOMISER have been treated as porous media. LTSH has been divided into two blocks i.e. LTSH strip and LTSH main in order to facilitate to show the direction of the porous medium.

**5.6 Porous Block Calculations**

**5.6.1 Inertial Resistance**

To analyze the flow through various tube modules like PSH, RH, LTSH, ECO, we assume it as a porous block .To balance the momentum loss, correct value of inertial resistance is given as input in FLUENT.

$$ \text{Inertial Resistance, (I.R.)} = \left( \frac{2 \times (P.D \times 9.81)}{\rho \times V^2 \times t} \right) \hspace{1cm} (67) $$

Where,

- \( P.D \) = Pressure drop in mm of water column across the module
- \( V \) = Velocity of fluid in m/s
- \( t \) = Thickness of porous block in meter
- \( \rho \) = Density in kg/m$^3$

The pressure drop obtained in module was crosschecked with input value the mass flow rate over the tube bundles. The mass flow rate should match with the
To develop a Predictive Tool for Boiler Tube Failure

expected mass flow rate of flue gases or air in case of CAVT. If it is not matching then the value of inertial resistance is adjusted to get desired pressure drop or mass flow rate by trial and error method.

Table 5.1: Initial calculations of inertial resistance for CAVT

<table>
<thead>
<tr>
<th>ZONE</th>
<th>P. D. (mmwc)</th>
<th>VELOCITY (m/s)</th>
<th>THICKNESS (m)</th>
<th>DENSITY (kg/m$^3$)</th>
<th>I.R. (1/m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>PSH</td>
<td>1</td>
<td>8</td>
<td>1.5577</td>
<td>1.225</td>
<td>0.1606566</td>
</tr>
<tr>
<td>RH</td>
<td>8</td>
<td>9</td>
<td>2.97182</td>
<td>1.225</td>
<td>0.53228641</td>
</tr>
<tr>
<td>FSH</td>
<td>6</td>
<td>12</td>
<td>0.84773</td>
<td>1.225</td>
<td>0.78721638</td>
</tr>
<tr>
<td>LTSH STRIP</td>
<td>0.5</td>
<td>13</td>
<td>0.2445</td>
<td>1.225</td>
<td>0.19380606</td>
</tr>
<tr>
<td>LTSH MAIN</td>
<td>16.5</td>
<td>13</td>
<td>4.545</td>
<td>1.225</td>
<td>0.34405373</td>
</tr>
<tr>
<td>ECONOMISER</td>
<td>17</td>
<td>14</td>
<td>2.0395</td>
<td>1.225</td>
<td>0.68113321</td>
</tr>
</tbody>
</table>

5.6.2 Porosity (Φ) Calculations

$\phi = \frac{\text{void volume in porous medium sample}}{\text{total volume of porous medium sample}}$

Note: Tube volume, void volume and total volume are calculated from Pro-E model of boiler.

Porosity is calculated for different zones as given in Annexure V

Porosity calculations for PSH

- Outer Diameter (mm) 51
- Thickness (mm) 4.5
- Inner Diameter (mm) 42
- Inner Cross Sectional Area (mm$^2$) 1385.622
- Outer Cross Sectional Area (mm$^2$) 2042.818898
- Tube Length 1 SET(mm) 135128
- Number of Coils 29
- Inner Circumference(mm) 131.94678
- Total inner surface area (mm$^2$) 517061430.1
- Total inner surface area (m$^2$) 517.0614301
- Outer Circumference (mm) 160.22109
- Total outer surface area (mm$^2$) 627860308
- Total outer surface area (m$^2$) 627.860308
- Tube Length Total (mm) 3918712
- Tube Volume Total (mm$^3$) 8005218927
- Projected Area (mm$^2$) 17142488.5
- Depth (mm) 12801.2
- Projected Volume (mm$^3$) 2.19444E+11

Porosity 0.963520518
Table No: 5.2 Zone and corresponding porosity of porous media

<table>
<thead>
<tr>
<th>Zone</th>
<th>Tube Volume (Solid) (mm$^3$)</th>
<th>Void Volume (mm$^3$)</th>
<th>Total Volume (mm$^3$)</th>
<th>Porosity (Φ)</th>
</tr>
</thead>
<tbody>
<tr>
<td>PSH</td>
<td>8.1E+9</td>
<td>2.11E+11</td>
<td>2.19E+11</td>
<td>0.9635</td>
</tr>
<tr>
<td>RH</td>
<td>3.91E+10</td>
<td>4.58E+11</td>
<td>4.97E+11</td>
<td>0.9214</td>
</tr>
<tr>
<td>FSH</td>
<td>1.2E+10</td>
<td>6.85E+10</td>
<td>8.05E+10</td>
<td>0.8512</td>
</tr>
<tr>
<td>LTSH STRIP</td>
<td>5.92E+09</td>
<td>2.54E+10</td>
<td>3.13E+10</td>
<td>0.8112</td>
</tr>
<tr>
<td>LTSH MAIN</td>
<td>659E+10</td>
<td>4.42E+11</td>
<td>5.08E+11</td>
<td>0.8703</td>
</tr>
<tr>
<td>ECONOMISER</td>
<td>403E+10</td>
<td>1.80E+11</td>
<td>2.20E+11</td>
<td>0.8169</td>
</tr>
</tbody>
</table>

5.7 Simulation of CAVT Using Fluent

The actual geometry of the flue gas path of the boiler created using Pro-E and meshed using GAMBIT is imported in FLUENT for analysis. The models and boundary conditions are given in the following discussion. The results obtained will then be compared with the experimental data of CAVT to validate the model.

5.7.1 Models and Boundary Conditions Used in CAVT Analysis

FLUENT (3d, pressure-based, k-epsilon)

<table>
<thead>
<tr>
<th>Model</th>
<th>Settings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Space</td>
<td>3D</td>
</tr>
<tr>
<td>Time</td>
<td>Steady</td>
</tr>
<tr>
<td>Viscous</td>
<td>k-epsilon realizable</td>
</tr>
<tr>
<td>Heat transfer</td>
<td>Disabled</td>
</tr>
<tr>
<td>Solver</td>
<td>Pressure based</td>
</tr>
<tr>
<td>Velocity formulation</td>
<td>Absolute</td>
</tr>
<tr>
<td>Porous formulation</td>
<td>Physical velocity</td>
</tr>
</tbody>
</table>

Material Properties

Air (fluid)

<table>
<thead>
<tr>
<th>Property</th>
<th>Units</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>kg/m3</td>
<td>1.225</td>
</tr>
<tr>
<td>Cp (specific heat)</td>
<td>J/kg-K</td>
<td>1006.43</td>
</tr>
<tr>
<td>Thermal Conductivity</td>
<td>W/m-K</td>
<td>0.0242</td>
</tr>
<tr>
<td>Viscosity</td>
<td>kg/m-s</td>
<td>1.7894001e-05</td>
</tr>
<tr>
<td>Molecular weight</td>
<td>kg/kgmol</td>
<td>28.966</td>
</tr>
</tbody>
</table>

Steel-tube (solid)

<table>
<thead>
<tr>
<th>Property</th>
<th>Units</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>kg/m3</td>
<td>7753</td>
</tr>
</tbody>
</table>
Cp (Specific heat) J/kg-K 486
Thermal conductivity W/m-K 35.38

<table>
<thead>
<tr>
<th>Condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Inlet</strong></td>
<td></td>
</tr>
<tr>
<td>Gauge total pressure (Pascal)</td>
<td>-49.04</td>
</tr>
<tr>
<td>Direction specification method</td>
<td>Normal to the boundary</td>
</tr>
<tr>
<td>Turbulence Specification Method</td>
<td>Intensity and Hydraulic diameter</td>
</tr>
<tr>
<td>Turbulent intensity (%)</td>
<td>2.362</td>
</tr>
<tr>
<td>Hydraulic Diameter (m)</td>
<td>12.75</td>
</tr>
<tr>
<td><strong>Outlet</strong></td>
<td></td>
</tr>
<tr>
<td>Gauge pressure (Pascal)</td>
<td>-529.67572</td>
</tr>
<tr>
<td>Backflow Direction Specification Method</td>
<td>Normal to Boundary</td>
</tr>
<tr>
<td>Backflow Turbulent Intensity (%)</td>
<td>2.14674</td>
</tr>
<tr>
<td>Backflow Hydraulic Diameter (m)</td>
<td>10.34944</td>
</tr>
</tbody>
</table>

**Solution Controls**

**Pressure Velocity Coupling:** SIMPLE

Under-Relaxation Parameters

<table>
<thead>
<tr>
<th>Variable</th>
<th>Relaxation parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure</td>
<td>0.3</td>
</tr>
<tr>
<td>Density</td>
<td>1</td>
</tr>
<tr>
<td>Body forces</td>
<td>1</td>
</tr>
<tr>
<td>Momentum</td>
<td>0.7</td>
</tr>
<tr>
<td>Turbulent Kinetic Energy</td>
<td>0.8</td>
</tr>
<tr>
<td>Specific Dissipation Rate</td>
<td>0.8</td>
</tr>
<tr>
<td>Turbulent Viscosity</td>
<td>1</td>
</tr>
</tbody>
</table>

**Discretization Scheme**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Scheme</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure</td>
<td>Standard</td>
</tr>
<tr>
<td>Momentum</td>
<td>QUICK</td>
</tr>
<tr>
<td>Turbulent Kinetic Energy</td>
<td>QUICK</td>
</tr>
<tr>
<td>Specific Dissipation Rate</td>
<td>QUICK</td>
</tr>
</tbody>
</table>

Plate 1 in Annexure VI shows the location of planes and points of readings used by the agency which performed the onsite CAVT. For obtaining velocity at various points across the pressure parts of the boiler imaginary planes and observation points were introduced in the geometry at different levels as given in the plate 1. Simulation was run for 1648 iterations and converged. Simulated CAVT results i.e. predicted velocity of cold air was appropriately tabulated and further compared with
To develop a Predictive Tool for Boiler Tube Failure

the experimental CAVT velocity readings. The simulated and experimental results are
given in the table. As porous media approach is used, velocity of air across the
pressure part was initially obtained as constant. But then the inertial resistance is
accordingly adjusted to get the desired mass flow rate through the pressure part.
Comparison of results is given in Table No. 5.3. Simulated results and actual results
are in acceptable variation.

Table: 5.3, Comparison of CAVT Results

<table>
<thead>
<tr>
<th>Reference plane</th>
<th>Plane Average velocity, m/s</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Experimental</td>
<td>Simulated</td>
</tr>
<tr>
<td>Top of LTSH Upper Bank</td>
<td>1.82</td>
<td>1.49</td>
</tr>
<tr>
<td>Bottom of LTSH Upper Bank</td>
<td>1.73</td>
<td>1.47</td>
</tr>
<tr>
<td>Top of LTSH Lower Bank</td>
<td>1.62</td>
<td>1.43</td>
</tr>
<tr>
<td>Bottom of LTSH Lower Bank</td>
<td>1.89</td>
<td>1.29</td>
</tr>
<tr>
<td>Top of Economizer</td>
<td>1.81</td>
<td>1.34</td>
</tr>
</tbody>
</table>

Difference in estimated and experimental values in the plane ‘Bottom of
LTSH Lower bank’ and ‘Top of Economizer’ is obvious because there is increase in
the velocity in the region and thus it is prone to erosion. This area must be attended to
find the cause of increase in velocity.

5.8 Discussion:

i) A result of Simulated CAVT plots shows constant velocity across the
pressure parts. This is because the inertial resistance is assumed as
constant along the depth of tube bundles for maintaining the uniform flow
rate of flue gases. The same concept is applied for CAVT.

ii) Measured velocity of air across the tube bundles is high as compared to
average velocity. The locations are prone to erosion failure.

iii) CAVT velocity contours shows that the velocity of air increase at the
bottom gap of primary super heater nose arch, re-heater bottom and at the
economizer bends.

iv) Past tube failure data also highlights that the tube failure due to erosion
had occurred more in this region.

v) Simulated CAVT results and experimental results are in acceptable
variation. Velocity variation trends match with the experimental results.
The geometrical model and the porous media approach is now can be used
for flue gas velocity and temperature prediction.