NUMERICAL AND EXPERIMENTAL MODELS OF FLOW IN THE CONVERGING SECTION OF A HEADBOX

Mohammad Reza Shariati **, Eric Bibeau*, Martha Salcudean** and Ian Gartshore**

*Process Simulations Limited (PSL), #204, 2386 East Mall, Vancouver, BC V6T 1Z3
**Department of Mechanical Engineering, University of British Columbia, Vancouver, BC

ABSTRACT

The present study investigates the flow through the converging zone of a headbox, both numerically and experimentally. The main focus of this study is on the turbulence quantities and secondary flows in the jet leaving the headbox. Both standard k-ε and non-linear k-ε turbulence models were used for the numerical simulation of the flow in the three-dimensional headbox model. A scale model of a hydraulic headbox was constructed from plexiglas and used with water as the working fluid. Laser Doppler Velocimetry (LDV) was used to measure mean and fluctuating components of the velocity at different locations in the model. Both numerical and experimental results from the present study show much smaller magnitudes for secondary flows, compared to those previously reported in the literature. It was also observed that the turbulence kinetic energy computed numerically is much higher than that measured experimentally.

1. Introduction

The pulp and paper industry is constantly challenged by complex and contradictory problems of how to produce higher quality paper from lower quality pulp with increased speed and reduced production costs. To address some of these issues this present study focuses on the analysis of the flow in the paper machine headbox.

In order to achieve good paper formation, fiber flocculation in the jet exiting the headbox must be avoided, and the fibers must be evenly distributed in the cross machine direction. Also cross machine velocities, which cause variations in the average fiber angle across the paper machine, should be minimized.

Recent studies on head boxes include those done by Seppo Syrjala et al [1] who used a commercial computer code (Phoenics) to study the flow behavior of a tapered manifold flow spreader used in a typical paper machine. This use of the k-ε model and an approximation of the channel as a slot opening at the side of the tapered manifold produced a simplified model of the flow. Gary Jones and Robert Minnow [2] used the FLUENT commercial computer code, which uses standard k-ε turbulent model, to study the simple Conver-flow and straight-channel headboxes. However, their assumption of uniform grid spacing, 2-dimensional flow, uniform cross direction profile and isotropic turbulence has limited the accuracy of their result in predicting the turbulence characteristics. Jari Hamalainen [3] studied the flow in the hydraulic headbox using the finite element method. Again the use of a linear k-ε model as a turbulence closure model has left the numerical model in some doubt for modeling the secondary flows in the headbox. The classic generation of secondary flows in the square duct due to anisotropy of the turbulence, has lead some researchers to believe that significant secondary flows are generated in the converging section of the headbox. It was also believed that these secondary flows have a large magnitude and extend over a large area in the exit plane, therefore having a significant adverse effect on the fiber orientation and paper quality. Most recently a computational study of the converging channel of the headbox by Aidun and
Kovacs\textsuperscript{[4]} using a nonlinear k-\(\varepsilon\) model has shown the existence of such large secondary flows in the headbox. Both fiber deflocculation capability and secondary flow generated in the headbox are direct outcome of the turbulent flow in the headbox. Therefore numerical simulation of the flow in the headbox depends directly on the turbulence model used in the simulation. Therefore in the present study, both standard k-\(\varepsilon\) and a nonlinear turbulence model have been used for flow simulation. The results from each of these models were compared with experimental data to estimate the accuracy and performance of each turbulence model.

2. Experimental Setup

To obtain experimental data, a physical model of the headbox using water as the working fluid, was designed and constructed. The model is shown in the figure (2). The model is approximately 1/5 scale of a typical headbox. The majority of the walls were built with 12-millimeter thick Plexiglas sheets to facilitate measurements with an Argon Ion Laser Doppler Velocimeter (LDV). In the scaled model, the tube bank before the headbox is replaced with two rows of diffusers with 20 diffusers in each row. This modification offers more control for simulation of different inlet condition. The flow rate through each individual pipe is measured and controlled separately. Uniform width of the exit opening from the headbox (the slice width) was ensured by careful adjustment of set screws located along the span of the exit. Measurements of the mean and fluctuating velocity components were done at several different locations in the headbox.

![Figure (1): Drawing of the headbox model. All dimensions are in cm.](image)

<table>
<thead>
<tr>
<th>Table (I): Major dimensions of the headbox model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spanwise Width of the model</td>
</tr>
<tr>
<td>Height at the inlet to converging section</td>
</tr>
<tr>
<td>Convergence ratio</td>
</tr>
<tr>
<td>Slice width</td>
</tr>
</tbody>
</table>

3. Mathematical Model
We consider here the steady state turbulent flow of a viscous, incompressible fluid with constant properties. The governing field equations are the Navier-Stokes and continuity equations, which are given by

\[
\frac{\partial u_i u_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \nabla^2 u_i
\]

\[
\frac{\partial u_j}{\partial x_j} = 0
\]

Where \( u_i \) is the velocity vector, \( p \) is the modified pressure (which can include a gravitational potential), and \( \nu \) is the kinematic viscosity of the fluid.

A usual time averaging process on the equation (1, 2) leads to the time averaged continuity equation and the Reynolds-averaged Navier-Stokes equation which are given, in Cartesian tensor notation as

\[
\frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} (\bar{\nu} \bar{S}_{ij} - \bar{u}_i \bar{u}_j)
\]

\[
\frac{\partial \bar{u}_j}{\partial x_j} = 0
\]

Where

\[
\bar{S}_{ij} = \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right)
\]

And

\[
\bar{u}_i \bar{u}_j = \tau_{ij}
\]

Is the Reynolds-stress tensor.

In these equations, the bar and prime indicate the time-averaged mean and the fluctuation from its mean value, respectively.

The effective viscosity formulation, which is a direct extension of the laminar deformation law, is given by

\[
\tau_{ij} = \nu_t \bar{S}_{ij} - \frac{2}{3} k \delta_{ij}
\]

where

\[
k = \frac{1}{2} \bar{u}_i \bar{u}_j
\]

\( k \) is the turbulent kinetic energy, and \( \nu_t \) denotes turbulent kinematic viscosity which, unlike its laminar counterpart, varies spatially, and is not a property of the fluid.

The most popular turbulence model for solving engineering problems is the standard \( k-\varepsilon \) model with the wall function, which was introduced by Launder and Splading \[5\]. This model introduces the transport equation for the two turbulence quantities, namely, the turbulence kinetic energy \( k \) and its rate of dissipation \( \varepsilon \). The \( k \) equation is :

\[
\frac{\partial}{\partial x_j} (k \bar{u}_j) = \frac{\partial}{\partial x_j} \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} + P - \varepsilon
\]

where

\[
P = -\bar{u}_i \bar{u}_j \bar{S}_{ij} \quad \text{and} \quad \varepsilon = \bar{\nu} \bar{S}_{ij} \bar{S}_{ij}
\]

\( P \) and \( \varepsilon \) are the rates of kinetic energy production and dissipation per unit mass, respectively. The transport equation for the kinetic energy dissipation is given in the form of
\[
\frac{\partial}{\partial x_j} \varepsilon \mathbf{u}_j = \frac{\partial}{\partial x_j} \left( \nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} + \left( c_1 P - c_2 \varepsilon \right) \frac{\varepsilon}{k} \quad (7)
\]

The isotropic viscosity assumption for the kinematic viscosity in terms of \( k \) and \( \varepsilon \), given by \( \nu_t = C_D k^2 /\varepsilon \), finally closes the system of equations 3, 4, 6, and 7.

The usual values of the constants are

\[
C_D=0.09, \quad C_1=1.44, \quad C_2=1.92, \quad \sigma_k=1.0
\]

\( \sigma_\varepsilon = \kappa^2 /[(C_2-C_1) C_D^{1/2}] \)

Where \( \kappa=0.41 \) is the Von Karman constant.

Other researchers in the past have demonstrated the wide range of applicability of this particular two-equation turbulence model by comparing its results with the available experimental data.

One of the major deficiencies of the standard \( k-\varepsilon \) model lies in its use of an eddy-viscosity model (equation 5) for the Reynolds-stress tensor. Eddy-viscosity models have two major problems associated with them:

- They are purely dissipative and hence cannot account for Reynolds-stress relaxation effect.
- They are oblivious to the presence of the rotational strains (e.g. they fail to distinguish between the physically distinct cases of plane shear, plane strain, and rotating plane shear).

In order to overcome these deficiencies, one has to use a nonlinear or anisotropic generalization of eddy-viscosity models. These models express the Reynolds-stress tensor as a higher order function of the mean velocity gradients. The model used in this study is that one suggested by Speziale [6]. One of the advantages of this model is its similarity to a standard \( k-\varepsilon \) model. It basically constitutes a two-equation model with an anisotropic eddy viscosity. The two equations are, as before, the transport equation of the turbulence kinetic energy \( k \) (eq.6) and dissipation rate \( \varepsilon \) (eq.7). The shear stress can be calculated from

\[
\tau_{ij} = \frac{2}{3} k \delta_{ij} - C_\mu \frac{K^2}{\varepsilon} S_{ij} - C_D C_\mu \frac{K^3}{\varepsilon^2} \left( S_{ik} S_{kj} - \frac{1}{3} S_{mn} S_{mn} \delta_{ij} \right) \\
- 2 C_\varepsilon C_\mu \frac{K^3}{\varepsilon^2} \left( \dot{S}_{ij} - \frac{1}{3} \dot{S}_{ij} \delta_{ij} \right)
\]

Where

\[
\dot{S}_{ij} = \frac{\partial S_{ij}}{\partial t} + U_k \frac{\partial S_{ij}}{\partial x_k} - \frac{\partial U_i}{\partial x_k} S_{kj} - \frac{\partial U_j}{\partial x_k} S_{ki}
\]

The accuracy of the new turbulence model was tested by a simulation of the flow in a square duct with side \( h=10 \) Cm and length \( L=100 \times h \). To insure grid independence, computations were carried out on 3 different grid sizes \( 9 \times 9, 11 \times 11, 15 \times 15 \) at each cross section and 50 grids along the length of the duct. By taking advantage of flow symmetry along the centerlines, actual computations were carried out only for a one quadrant of the duct. Figure (2) shows the direction and magnitude of the secondary flows at \( X=80 \times D_h \) and \( \text{Re}_d=250000(D_h \text{ is the hydraulic diameter and } \text{Re}_d \text{ is the Reynolds number based on } D_h) \).

There is a good agreement between present calculation the numerical predictions by Myong and Kobayashi [7] and the experimental data reported by Gessner and Emery [8].
4. Results and Discussion

The geometry and boundary conditions for the numerical calculations are same as those used by Aidun et al at [4]. Figure (3) shows the computational geometry. Dimensions and boundary conditions are given in table (2).

Table (II): Computational geometrical basic dimensions.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length in X direction</td>
<td>L_X=1.125 m</td>
</tr>
<tr>
<td>Length in Y direction</td>
<td>L_Y=3.75 m</td>
</tr>
<tr>
<td>Height at inlet in Z direction</td>
<td>L_Z=0.375 m</td>
</tr>
<tr>
<td>Contraction ratio</td>
<td>10</td>
</tr>
</tbody>
</table>

Figure (2): Secondary velocity profile along the wall bisector. Experimental data from Gessner et al; computational results by Myong et al (Y*=Y/h, Z*=Z/h, U_b is the bulk velocity in the streamwise direction)
Figure (3): Computational geometry.

Table (III): Boundary condition for numerical calculations.

<table>
<thead>
<tr>
<th>Inlet</th>
<th>Uniform velocity $U_{in} = 1.22$ m/s</th>
</tr>
</thead>
<tbody>
<tr>
<td>Exit</td>
<td>Gradient Boundary condition</td>
</tr>
<tr>
<td>Top</td>
<td>Wall</td>
</tr>
<tr>
<td>Bottom</td>
<td>Wall</td>
</tr>
<tr>
<td>South($Y=0.0$)</td>
<td>Symmetry</td>
</tr>
<tr>
<td>North($Y=L_Y$)</td>
<td>Wall</td>
</tr>
</tbody>
</table>

The mean component of the velocity in the X direction ($U_{m}$) is shown in figure (4). Data are selected along the line joining the center of the inlet plane to the center of the exit plane at the symmetry plane. Results from the nonlinear model are closer to the experimental data than those obtained using standard $k-\varepsilon$ model.
The mean velocity component in the Y direction (V or cross direction velocity) at the exit plane is shown in figure (4). Results from standard k-ε model and the nonlinear model are very close to each other and comparable to experimental data closer to the wall. Deviation of the experimental data from computed values close to the centerline is probably due to a deflection of the plates due to the fluid pressure. If this CD velocity is due to turbulence anisotropy, and has the same origin as the secondary flows in the square duct, then one can assume they should have the same pattern and magnitude. That is, fluid should move towards the corner along the corner bisector and move towards the center along the side wall bisector. Thus the CD velocity along the wall bisector should always be negative. Also secondary flows in a square duct reach their maximum value of approximately 2 percent of the bulk velocity only in the fully developed region and are much smaller in magnitude in the developing region. In the view of these observations, the CD velocity magnitude and direction calculated by the nonlinear model looks like he most accurate one. Unfortunately due to high ratio of the signal to noise ratio very close to a wall reliable experimental data could not be obtained in the region close to the side wall. The numerical results of Aidun and Kovacs [4] appear to be quite incorrect.
Figure (4): CD velocity at the exit plane $Y^*=Y/L_y$, $U_{in}$ is the bulk velocity in the streamwise direction at the inlet.

Figure (5) shows rms values of the fluctuating components of the velocity in the X and Y direction $u'$ and $v'$ normalized with the mean component of the velocity in the X direction ($U$). Figure (6) the rms data are normalized by the value of $u'$ at the inlet ($u'_{o}$). Rms values of the fluctuating component of the velocity in the X direction $u'$ diminishes faster than $v'$ at the beginning of the contraction but later both fluctuating components increase in magnitude and become almost equal in magnitude near the end of the headbox.

Figure (5): Rms. values of the fluctuating component of velocities $u'$, $v'$ normalized by $U$ (local mean velocity in the X direction).
Figure (6): $u', v'$ normalized with the $u'$ at the inlet ($u'_o$)

Figure (7): Turbulence Kinetic energy, $k^* = k/k_{in}$. Here $k_{in}$ is the turbulent kinetic energy at the inlet to the headbox.

In figure (7) turbulence kinetic energy $k$ at the symmetry plane as calculated by two different turbulence models, is shown along with the experimental values. In calculating the experimental values for $k$, the third component of the fluctuating velocity ($w'$) was taken as an average of the first two fluctuating components $u'$ and $v'$. Although the nonlinear turbulence model gives a better result, values from both
standard k-ε and nonlinear k-ε are grossly exaggerated. In the case of these flows with such a high rate of strain, the generation term in the k equation is apparently much too large\cite{9}.

5- CONCLUSION AND SUMMARY

Mean and fluctuating components of the fluid velocity in the headbox model were measured experimentally. The experimental results were compared with numerical data obtained using two different turbulence models and the following observations are made.

1) Computational and experimental results show that secondary flows in the headbox are much smaller in magnitude than those reported by other researchers. Also these secondary flows are confined to a very small region near the corners of the headbox.

2) Both standard and nonlinear turbulence models predict mean velocity distributions reasonably well. The nonlinear model is generally closer to measured values.

3) Although the nonlinear k-ε turbulence model produces better results than the standard k-ε model, both models fail to predict accurate values for the turbulence kinetic energy.

Overall, we concluded that the calculated mean velocity components are reasonably well predicted, but that the fluctuating components require a better turbulence model for accurate prediction.

ACKNOWLEDGMENTS

The authors gratefully acknowledge funding from the Natural Sciences and Engineering Research Council of Canada and Forest Renewal British Columbia (FRBC).

Reference
