ABSTRACT
The turbulent flow in a hydraulic headbox has been numerically studied. The flow velocities, pressure, kinetic energy, and dissipation in the manifold, diffuser tubes and converging section have been simultaneously calculated. Here, we present the numerical results inside the headbox, especially the flow inside the converging section resulting from the interaction of tube jets. Results are presented in the form of machine direction (MD), cross-machine direction (CD) components of velocity and also turbulent quantities at different sections along the length of the converging section. The present study is part of a larger effort to develop advanced computer models for predicting complex flows in pulp and paper headboxes, and serves as an important step for predicting fluid-fiber interactions to provide paper manufacturers better control over fiber orientation, fiber distribution, and sheet properties.

INTRODUCTION
Hydraulic-type headboxes are commonly used in the pulp and paper industry. Stock is admitted at the large end and flows across the width of the manifold to the small end. A portion of the stock is recirculated to prevent a pressure build-up at the manifold exit. The tube bank connects the manifold to the converging section, which produces a free jet that impinges on the forming section. In order to supply well dispersed stock containing a constant percentage of fibers to all areas of the sheet-forming section, and because fibers in headboxes tend to form flocs rapidly, removal of flow non-uniformities and the creation of high intensity turbulence are required in headbox designs. Variation of fiber orientation and basis weight profiles in the cross-machine direction are dependent on the headbox design and operation mode. Eventually, headbox designs may be flexible enough to provide paper manufacturers with the ability to select sheet properties they require with minimal changes to the headbox. This will require a better control of fiber distribution emanating from the headbox, a method to control MD/CD ratios over a wide range, and the prevention of flow non-uniformities originating in the headbox in the machine direction and cross-machine direction. To achieve this goal, a detailed understanding of fluid flow within the entire headbox and fluid-fiber interaction is required.

The analysis and design guidelines to obtain a uniform flow distribution at the converging section exit have been previously formulated [1]. Until recently, the complexity of the geometry and the three-dimensional turbulent flow field occurring in headboxes did not allow for a complete flow calculation. Therefore, there have been few numerical calculations of flows in headboxes. Jones and Ginnow [2] calculated flow parameters in a straight section diffuser in three dimensions and in a Beloit experimental headbox in two dimensions. Predictions compared favorably with available experimental data. The authors recommended further validation of the parameters in the $k-\epsilon$ model. Shimizu and Wada [3] calculated a generic headbox using a shape-fitted coordinate system. The flow distribution in the manifold was investigated in two dimensions and the converging section was calculated in three dimensions with assumptions of periodicity. The jets from the diffuser tubes were modeled in three dimensions using calculation results obtained for a single tube. Hämäläinen [4] linked two-dimensional models of the manifold, the rectifier section, and the converging section using finite element. The pressure drop in each tube was assumed to be that of a single tube using the homogenization technique. Separate three-dimensional models of the manifold and of the converging section have been performed by Lee [5] to investigate various effects of headbox control devices on flow characteristics and fiber orientation. Bandhakavi and Aidun [6] studied the flow characteristics through a simplified tube block and the converging section by using different turbulence model. Studies investigating secondary flows in the converging section [7, 8] have also been performed, where it was recommended that higher order versions of the turbulence model may be required.

The above studies have known limitations. The diffuser tubes were either ignored, modelled in two-dimension, or assumed to have single tube behavior. These models cannot account for the flow non-uniformities existing across the diffuser tubes. Our previous work has been concentrated on obtaining the flow distribution through the diffuser tubes [9]. The emphasis of this work is to predict the flow in the converging section. The object of this study is to demonstrate the predictive capability of a three-dimensional headbox model which can be used to trouble-shoot existing headboxes, evaluate proposed retrofits, compare headbox designs, predict the influence of control devices and operation modes on flow behavior, and serve as an important step for predicting fluid-fiber interactions.
HEADBOX NUMERICAL MODEL

The three-dimensional incompressible Reynolds averaged Navier-Stokes equations are solved. Turbulence closure is obtained by the use of the standard \( k - \epsilon \) model with the wall function treatment. A domain segmentation method in conjunction with curvilinear grids is used in the present study. The headbox is decomposed into the manifold, diffuser tubes and the converging section. A solution is obtained by repeatedly applying a single-domain solution solver to all the segments, and cycling through all the segments until the residuals are sufficiently small. Efficient communication between the segments is established to achieve fast convergence by appropriately transferring data between adjacent segments for each iteration. Validation cases for manifold applications can be found in Reference [10]. The detailed formulation of the code, developed at the University of British Columbia, can be found in Reference [11]. The accuracy and performance of the standard \( k - \epsilon \) model and nonlinear turbulence models were studied and compared with experimental data in Reference [12].

HEADBOX PHYSICAL MODEL

The headbox geometry used in this study is shown in Figure 1. It has a linear rectangular tapered manifold, a tube bank which consists of 120 rows of tubes and 8 columns giving a total of 960 tubes and a symmetrical converging section. The headbox manifold consists of a tapered rectangular duct 4000-mm wide with the largest rectangular cross-section measuring 600 mm by 266 mm and the smallest 152 mm by 266 mm. The inlet duct diameter is 457 mm and the outlet duct diameter is 86 mm. Each tapered diffuser tube is 275-mm long with an inlet square cross-section 13 mm in width and an outlet square cross-section 31 mm in width. The square outlet diffuser tubes are separated by a 2 mm wall on all sides. There are 8 rows of 120 diffuser tubes comprising the turbulence generating section connecting the manifold to a symmetrical converging section 760-mm long.

Taking advantage of flow symmetry, only half of the flow domain is calculated: half of the manifold duct, the first four rows of diffuser tubes, and half of the converging section. The grid used contains 544 \( \times 13 \times 13 \) grid nodes for the manifold, 7 \( \times 7 \times 9 \) grid nodes for each diffuser tube, and 961 \( \times 33 \times 13 \) for the converging section. The grid is generated using a combination of an elliptic grid generation method and an algebraic generation grid method.

Different types of boundary conditions are used: inlet velocity for the headbox entrance, outlet velocity to model the recirculating flow; zero-slip wall condition to model all headbox internal surfaces; symmetric boundary conditions with zero flux and a free-slip condition to model the symmetry plane; and an imposed shear-stress caused by the area change of the converging section at the converging section outlet. Results were obtained using water and an inlet manifold flow velocity of 5.8 m/s.

RESULTS

The results for the manifold flow distribution and flow non-uniformity in diffuser tubes are presented in Reference [9]. Because the flow exiting the diffuser tubes is highly non-uniform and we are eventually interested in determining the fiber distribution...
Figure 2: MD velocity in the cross-machine direction in the converging section

Figure 3: MD velocity in the converging section
and orientation, it is important to properly model the correct flow conditions entering the converging section and the turbulence characteristics of the flow.

Figure 2 shows the MD velocity (non-dimensionalized by the local averaged MD velocity) at different planes along the MD direction. The non-uniformity from the diffusers exit is carried forward through the converging section as shown in the left frames. Flow of an oscillatory nature can be observed at the entrance to the converging section. The oscillations caused by the diffusers are still visible at the slice exit but they are quite small as shown in the right frame. The MD velocity variation is significant. The converging section would not address flow non-uniformity originated from poor manifold flow, but the structures resulting from the tubes do not survive as they accelerate along the converging section. The 3D view of the converging section MD velocity is presented in Figure 3.

Figure 4 shows the CD velocities caused by the tube bank decrease as the flow approaches the converging section exit. However, there are some flows in the cross-machine direction of the order of percents of the MD flows which may be traceable to the manifold. The CD velocity here is also non-dimensionalized with the local averaged MD velocity.

The turbulence intensity is important for de-flocculation and fiber orientation. One can observe from Figure 5 that despite the increase in the velocity, the turbulence intensity drops along the converging section. The ability to properly predict the correct turbulence in this region is limited by the time-averaging of the momentum equations. The turbulence intensity drops markedly through the first section of the converging section and then increases somewhat as it approaches the exit. The average length scale is presented in Figure 6 and the elongation factor which is important for the fiber-fluid interaction is shown in Figure 7.

CONCLUSIONS

A computational model is presented for the prediction of flow characteristics in a complete hydraulic headbox. The model uses block-structured curvilinear grids to allow the treatment of the complex geometry occurring in headboxes. The results show that the converging section effect is dominated by the contraction ratio. The converging section eliminates most of the structures generated by the diffusers. However, the non-uniformities originating from the manifold are not smoothed out. The turbulence drops considerably as a result of the accelerating flow and the turbulence might not be sufficient to break up the flocs.

ACKNOWLEDGMENTS

The authors would like to thank Weyerhaeuser for the financial support. Funding from the Forest Renewal British Columbia (FRBC), Natural Sciences and Engineering Research Council of Canada and from Process Simulations Limited (PSL) is gratefully acknowledged.
Figure 5: Turbulence intensity in the cross-machine direction in the converging section

Figure 6: Averaged length scale in the cross-machine direction in the converging section
References


