An investigation of boundary layer growth in pulp suspensions over a flat plate

Emelie Wibron
2013-05-28
Abstract

The pulp and paper industry consumes high amount of energy when producing paper. Large flows of pulp suspensions are handled during the process. There is still a lack of knowledge of the flow and fluid properties of pulp suspensions which is important in the design and operation of more efficient process equipment. The boundary layer growth and near wall flow over a flat plate in pulp suspensions were investigated in this thesis using Computational Fluid Dynamics (CFD) simulations. The numeric results were compared with experimental results obtained by Claesson et al (2013). The pulp suspension flow was simulated as a Bingham plastic fluid using different values of the yield stress. A variation of the inlet flow velocity was also conducted. Both the velocity profiles and the pressure drops were studied. A comparison of the boundary layer growth in the CFD simulations and the experiments showed that the velocity profile was established faster in the experiments and that the boundary layer thickness was thinner. An increased value of the yield stress gave a more similar velocity profile. The experimental values of the pressure drops were lower than the ones obtained in the CFD simulations. A lower value of the yield stress gave a more realistic pressure drop. This is a contradiction since lower values of the yield stress gave a worse agreement of the velocity profiles. Therefore, the wall treatment for the pulp suspension is suggested to be revised as well as the rheology of the fluid model.
## Contents

1 Introduction ......................................................... 1  
  1.1 Background .................................................... 1  
  1.2 Objectives .................................................... 1  

2 Theory  .............................................................. 3  
  2.1 Equations of fluid dynamics  ................................. 3  
  2.2 Boundary conditions ........................................... 4  
  2.3 Rheology of pulp suspension .................................. 4  
  2.4 Flow regimes ................................................... 6  
  2.5 Boundary layer .................................................. 6  
  2.6 Computational Fluid Dynamics  ............................... 7  

3 Numerical method .................................................. 9  
  3.1 Geometry ....................................................... 9  
  3.2 Mesh ............................................................ 10  
  3.3 Boundary conditions ........................................... 11  
  3.4 Simulated cases ................................................ 11  

4 Results  .............................................................. 13  
  4.1 Boundary layer growth ......................................... 13  
  4.2 Velocity profiles ............................................... 18  
  4.3 Pressure drop ................................................... 20  

5 Conclusions .......................................................... 23  

References ............................................................ 25
1 Introduction

1.1 Background

The worldwide pulp and paper industry consumes high amount of energy which could be reduced with a better understanding of the underlying physics. One of the main things that need to be further investigated is the flow and behaviour of pulp suspensions, its rheology and how it interacts with walls. Computational Fluid Dynamics (CFD) can be a helpful tool to provide this knowledge which would be important in the design and operation of more efficient process equipment. A decrease in energy demand would both increase the profit in the industry and save the environment.

The pulp and paper industry uses mostly wood as raw material when producing paper. The raw material, logs, is divided into small pieces, wood chips, and the individual fibres are separated from each other either by chemical or mechanical processes. The fibres are then treated with different chemicals in order to give the pulp the desired properties suited for the specific use of the final paper. The pulp is washed, cleaned and further bleached to give the final paper its brightness. The pulp is finally dried.

The consistency of the flowing pulp suspensions varies mostly in the range of 0.5 to 15% by mass during the pulping process. Fibre networks are formed already at very low consistencies, below 1% by mass, which makes the rheology of the fluid more complicated. The flow and fluid properties of pulp suspensions have been studied by several researchers, but there is still no general description of the flow structure. Therefore there are no standard models developed to be used in CFD simulations.

1.2 Objectives

The objective of this thesis is to investigate the boundary layer growth and near wall flow over a flat plate in non-Newtonian fluids in general and in pulp suspensions in particular. CFD simulations are carried out in the commercial software ANSYS CFX 14.0 and the numeric results are compared to the experimental results obtained by Claesson et al (2013).


2 Theory

2.1 Equations of fluid dynamics

The governing equations of fluid dynamics are the conservation laws for mass, momentum and energy. These laws can be stated in both differential and integral form. The first equation results from applying mass conservation to a fluid flow and is called the continuity equation

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \]  

(1)

where \( \rho \) is the fluid density and \( \mathbf{u} \) is the velocity vector of the fluid flow. The first term represents the rate of increase of density and the second term represents the rate of mass flux passing out of the volume. The next equation results from applying Newton’s Second Law to a fluid flow. It yields the Navier-Stokes equation

\[ \frac{\partial}{\partial t}(\rho \mathbf{u}) + \mathbf{u} \cdot \nabla (\rho \mathbf{u}) = -\nabla p + \mu \nabla^2 \mathbf{u} \]  

(2)

which is a vector equation, also known as the transport equation of momentum. This form of the equation considers a Newtonian fluid. The first two terms represent the rate of increase of momentum and convection of momentum. The last two terms represent pressure and viscous forces where \( p \) is the pressure and \( \mu \) is the dynamic viscosity. The last term in the equation is rearranged when a non-Newtonian fluid is considered and the constant viscosity is changed for the effective viscosity. The last conservation equation is the energy equation

\[ \frac{\partial E}{\partial t} + \nabla \cdot (E \mathbf{u}) + \nabla \cdot (\rho \mathbf{u}) = 0 \]  

(3)

where \( E \) is the total energy per unit volume, \( \mathbf{u} \) is the velocity vector and \( p \) is the pressure. The terms from left to right represents rate of total energy change, total energy flow and rate of work of pressure forces. This equation considers a fluid with no viscosity. Rate of work of viscous forces and rate of heat added are terms that should be included in a more general case. In addition to the conservation laws, a thermodynamical equation of state gives the pressure as a function of other thermodynamic variables in order to be able to solve any fluid dynamic problem.

Depending on the type of problem it might be possible to simplify all these equations. For example incompressible flow leads to a simplification since the derivatives of the density can be cancelled. But even under these simplifications are Navier-Stokes equation still non-linear and have seldom an analytic solution. Therefore numerical methods have to be applied to solve and analyse flows by using CFD. The solution yields a velocity field which can be used to find other quantities of interest.
2.2 Boundary conditions

The described governing equations of fluid dynamics are subject to boundary conditions in order to completely describe any problem. There are two different types of boundary conditions. A Dirichlet boundary condition defines the value of a variable at the boundary while a Neumann boundary condition defines the gradient normal to the boundary of a variable at the boundary. Different types of boundary conditions can be used for different variables. In some flows there is a plane of symmetry which is the case if the geometry is symmetric about the plane and the velocity field is the same on both sides. Normal velocity and normal gradients of all variables equal to zero are then defined at the plane of symmetry.

There are many different boundary conditions that may be applied to the inlet and the outlet of a domain. Velocity conditions define the velocity vector of the flow which is useful when the velocity profile for an incompressible flow is known at the inlet or at the outlet. A mass flow inlet condition is more appropriate if the flow is compressible. A pressure condition is useful when neither the flow rate nor the velocity is known and can be applied both to compressible and incompressible flows. Boundary conditions are also applied at the interfaces between a fluid and a rigid surface. A no-slip condition is defined at the walls in viscous flows which means that the tangential fluid velocity is equal to the velocity of the wall and the normal velocity component is defined to be zero.

2.3 Rheology of pulp suspension

Rheology is the science of deformation and flow of materials and describes all different kinds of materials, ranging from solids to fluids. Different rheology models are valid for different fluids, depending on the fluid properties. Fluids that have a linear relationship between the shear stress and the shear rate

\[ \tau = \mu \dot{\gamma} \]  

(4)

where \( \tau \) is the shear stress and \( \dot{\gamma} \) is the shear rate are called Newtonian. The constant of proportionality is the viscosity which in this case is independent of the shear rate. The viscosity can be thought of as a measure of a fluid’s resistance to flow. A fluid with high viscosity resists motion more than a fluid with low viscosity. This is due to internal friction in the fluid. Non-Newtonian fluids have a non-linear dependence between the shear rate and the shear stress which means that the viscosity is dependent of the shear rate. Non-Newtonian fluids are classified according to this dependence which is illustrated in the rheogram in Figure 1. Fluids for which the shear stress increases less than proportional to the shear rate are called shear thinning fluids while fluids for which the shear stress increases more than proportional to the shear rate are called shear thickening.
Some non-Newtonian fluids act like solid materials until the shear stress exceed a specific level called the yield stress. These materials start to flow when the shear stress overcome the yield stress. The yield stress is the single most important characteristic of the rheology of pulp suspensions but shear thinning behaviour has also been observed according to Wikström (2002). A pulp suspension flow is a two-phase flow consisting of fibres and water. The fibres tend to aggregate and form coherent networks above a certain concentration level which makes the flow behaviour very complex. A two-phase flow can be modeled as a one-phase flow by using a rheology model for the effective viscosity. The reason for using a rheology model instead of modeling the fibres and the water separately is the complexity of the pulp suspension flow. Pulp suspensions are often modelled with a non-Newtonian Bingham plastic rheology model expressed as

\[ \tau = \tau_y + \mu_p \dot{\gamma} \]  \hspace{1cm} (5)

where \( \tau \) is the shear stress, \( \tau_y \) is the yield stress, \( \mu_p \) is the so called "plastic" viscosity and \( \dot{\gamma} \) is the shear rate. The yield stress can be measured experimentally with a rheometer when the Bingham plastic model is used to model pulp suspensions. A sample is placed inside the rheometer and the yield stress is measured as the force needed to generate movement in the fluid while being in rest at the beginning. Shear thinning and shear thickening fluids are often modelled with the simple power law model

\[ \tau = K \dot{\gamma}^n \]  \hspace{1cm} (6)

where \( K \) is a consistency coefficient and \( n \) is a flow behaviour index.
2.4 Flow regimes

The structure in a fluid flow depends on the relation between the inertial forces and the viscous forces. This is described by the dimensionless Reynolds number

\[ Re = \frac{\text{inertial forces}}{\text{viscous forces}} = \frac{\rho u L}{\mu} \]  

(7)

where \( \rho \) is the density, \( u \) is the velocity, \( L \) is the characteristic length and \( \mu \) is the viscosity. A lower Reynolds number means that the viscous forces dominate in the flow and that the laminar flow regime is valid. Laminar flows are rather smooth with well-defined layers. The turbulent regime is valid at a higher Reynolds number. Flows in this regime are irregular and unpredictable. A transition regime is found between the two regimes in which both laminar and turbulent properties can be observed. The transition takes place at different values of the Reynolds number depending on the flow geometry. For pipe flow the transition to turbulence takes place at around 2300.

2.5 Boundary layer

The no-slip boundary condition for a viscous fluid requires that the velocity at a solid wall vanishes entirely. The layer where the velocity decreases to zero is called the boundary layer. The thickness of the boundary layer approaches zero as viscosity goes to zero. The velocity in the boundary layer smoothly joins that of the outer flow which makes the definition of the boundary layer thickness rather arbitrary. One measure of the boundary layer thickness is the distance from the wall to the position where the velocity is 99% of the velocity in the fully developed flow. That is where \( u = 0.99U \) if \( u \) is the velocity and \( U \) is the velocity far from the plate. This will be denoted \( \delta_{99} \). The boundary layer thickness of a Newtonian fluid flow over a flat plate can be described by the approximative Blausius solution

\[ \delta_{99} = 4.9 \sqrt{\frac{\mu x}{\rho U}} \]  

(8)

where \( \mu \) is the viscosity, \( x \) is the distance from the edge of the plate and \( \rho \) is the density. According to this equation an increase in viscosity or decrease in velocity far from the plate will lead to an increase of the thickness of the boundary layer. The Blasius solution agrees with experimental data until the local Reynolds number is larger than some critical value. The critical Reynolds number varies but for a boundary layer of a fluid flow over a flat plate it is found to be about \( 10^6 \) according to Kundu and Cohen (2000).
2.6 Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) uses numerical methods in order to predict fluid flow phenomena based on the governing equations of fluid dynamics. The most used numerical methods is based on the finite volume method in which the whole domain is divided into small cells. Defined conditions are usually the flow geometry, the physical properties of the fluid and the boundary and initial conditions of the flow field. The transport equations are integrated over the cells into a system of algebraic equations which are then solved. The flow of non-Newtonian fluids can be solved for by changing the viscosity for the effective viscosity. The transport equation of momentum is then expressed as

\[
\frac{\partial}{\partial t}(\rho u) + u \cdot \nabla(\rho u) = -\nabla P + \nabla(\mu_{eff} \nabla u)
\]

where the effective viscosity for a Bingham plastic fluid in a simple shear flow is defined as

\[
\mu_{eff} = \frac{\tau_y}{\frac{\partial u}{\partial y}} + \mu_{\infty}
\]

where \(\tau_y\) is the yield stress and \(\mu_{\infty}\) is the viscosity at high shear rate.

The use of CFD as an engineering tool has increased parallell to the development of computer power during the past decades. When using CFD one hopes to get an approximate solution as close as possible to the exact one. But the results obtained by CFD are never completely exact. There are many sources of error involved in the predictions where the most common ones are discretization errors, input data errors, initial and boundary condition errors and physical model choice. The solution is said to be convergent if it approaches the exact solution as the grid spacing, control volume size or element size tend to zero. However, a convergent solution does not eliminate all the possible sources of error.
3 Numerical method

3.1 Geometry

The geometry was originally a squared pipe with a length of 380 mm and a cross section area of 40×40 mm. The pipe in the model used in the CFD simulations was extended at the back to a length of 480 mm. This change was made to decrease the effect of the outlet boundary condition. At the bottom of the squared pipe there was a plate with a length of 220 mm, width of 40 mm and thickness of 10 mm. The plate was placed 80 mm from the inlet boundary. The first 10 mm of the plate was chamfered with an angle of 45 degrees to minimize the recirculation formed in front of the plate when running high viscous fluid flows at low velocities. To reduce the computational time a symmetry condition was used at z = 20 mm and the model of the geometry had therefore a width of just 20 mm. The origo of the coordinate system was located at the left side of the leading plate edge. The total model of the geometry and the mesh of it is depicted in Figure 2.

![Figure 2: The geometry and the mesh.](image-url)
3.2 Mesh

The mesh contained approximately 240,000 elements with a higher density of elements near the plate and the walls. Several meshes, in both two and three dimensions, with various cell densities were tested. The two dimensional models saved a lot of computational time but a three dimensional model turned out to be necessary in order to take the wall effects into account properly. Therefore an extruded two dimensional mesh consisting of prisms and hexahedrons was used. Since the boundary layer was of most interest it was important to capture the velocity profile in the near wall region correctly. Inflation layer meshing techniques turned out to be an important tool to decrease mesh size and increase numeric quality. It was applied both at the top and the bottom of the mesh. The first layer height at the bottom was 0.05 mm and the growth rate was set to 1.1 with a maximum of 30 layers. The first layer height at the top was 0.5 mm with a maximum of 10 layers while the growth rate was unchanged. A close up of the mesh where this is more visible is illustrated in Figure 3. The overall body sizing of the mesh was 3 mm.

![Figure 3: Close up of the mesh.](image)
3.3 Boundary conditions

The inlet boundary condition was taken as a velocity and the outlet boundary condition as a static pressure. Different velocities were used at the inlet while the boundary condition at the outlet remained the same. A no-slip condition was used at the plate and the walls. A symmetry boundary condition was set at $z = 20$ mm.

3.4 Simulated cases

The pulp suspensions were modeled with a non-Newtonian Bingham plastic model. The yield stress was measured in the experiment performed by Claesson et al (2013). Pulp suspensions at a consistency of 0.5% turned out to have a measured yield stress of 55 Pa (95 and 157 Pa for 1.0 and 1.8% respectively). CFD simulations were performed with these values of the yield stress. A yield stress of 10 Pa was also simulated since Nikbakht et al (2013) have obtained results that indicate that the previously mentioned values of the yield stress could have been overestimated. According to them the yield stress of 10 Pa correspond to 1% consistency of the same pulp type used by Claesson et al (2013). The three different average velocities used above the plate were 0.24, 0.48 and 0.72 m/s corresponding to an inlet velocity of 0.18, 0.36 and 0.54 m/s. All possible combinations of the yield stress and the velocity have been studied.

If the simulated fluid had been water, the Reynolds number would have been at least equal to approximately

$$Re = \frac{1000 \times 0.24 \times 0.03}{1 \times 10^{-3}} = 7200$$

since this was the lowest value that could be obtained for the given density, characteristic length and viscosity. Therefore, the flow regime would have been turbulent if water was used. However, this was not the case for the pulp suspension flow which has a much higher viscosity. The effective viscosity, Equation 10, was used to estimate the Reynolds number for pulp suspension flow. The minimum of the effective viscosity was equal to approximately

$$\mu_{eff} = \frac{10}{0.72} + 0.1 \approx 0.3$$

and therefore, the Reynolds number became no larger than approximately

$$Re = \frac{1000 \times 0.72 \times 0.03}{0.3} = 72$$

which means that the flow was in the laminar regime.
4 Results

4.1 Boundary layer growth

The boundary layer growth was compared for different values of the yield stress and the velocity at different positions to determine when the velocity profile was fully developed. The boundary layer growth at constant position in z-direction but different positions in x-direction from the plate edge is depicted in Figure 4.

![Figure 4: Boundary layer growth along the length of the plate in the simulations.](image)

The constant position in the z-direction was equal to 3.5 mm from the wall which was the value used by Claesson et al (2013). Figure 4 is an illustration of the CFD simulation of a velocity equal to 0.48 m/s and an yield stress equal to 95 Pa, the simulations of other values showed a similar boundary layer growth. It can be seen that the velocity is not close to fully developed until somewhere at between \(x = 20\) and \(x = 30\) mm. An illustration of the same case as in Figure 4 but for the whole height of the pipe is found in Figure 5. The experimental results obtained by Claesson et al (2013) indicate that the velocity profile was fully developed already at \(x = 20\) mm. This is illustrated in Figure 6 for a velocity equal to 0.48 m/s and an yield stress equal to 95 Pa, but a fully developed velocity at \(x = 20\) mm was observed independent of both velocity and measured yield stress.
Figure 5: Overview of the simulated velocity profiles in the whole height of the pipe.

Figure 6: Experimental boundary layer growth along the length of the plate according to Claesson et al (2013).
The velocity profiles in the CFD simulations can also be shown as contour plots to give an overview in three dimensions as depicted in Figure 7. The wall turned out to have a very large effect because of the no-slip condition that was used. This could be seen by studying the boundary layer growth at different distances from the wall. The boundary layer growth at a constant position in x-direction of 30 mm but different positions in z-direction is illustrated in Figure 8. At $z = 3$ and $z = 6$ mm the velocity profiles were clearly not fully developed yet while at $z = 15$ and $z = 18$ mm they were close to be. The wall seemed to have a much larger effect in the CFD simulations compared to the experiments which indicate that the wall treatment used in the CFD simulations was overestimating the wall shear stress.

In reality, the pulp suspension flow is a two-phase flow and not homogenous as the CFD simulations assumed. Coherent networks are formed in a pulp suspension flow as a result of the large ratio between the length and the diameter of the fibres. The concentration of fibres are therefore higher in some parts of the pulp suspension flow and these areas are usually called flocs. According to Fock (2009) there is a decreased amount of fibres near the walls in a pulp suspension flow. The layer closest to the wall could therefore have a lower network strength than the wall model applied. Another impact of the flow behaviour close to the wall is the fact that when the shear stress has exceeded the yield stress it might decrease along the streamline according to Wikström (2002).
A comparison to the Blasius solution for a Newtonian fluid was of interest in order to illustrate the differences of Newtonian boundary layer growth and the boundary layer growth of pulp suspensions as depicted in Figure 9. It could be seen that the boundary layer thickness of the pulp suspensions increased with an increase in velocity or a decrease in consistency in contrast to the Newtonian boundary layer development described by the Blasius solution. The boundary layer thickness obtained by the Blasius solution increases with a decrease in velocity or an increase in viscosity. The same conclusion was made by Claesson et al (2013) based on their experimental results.

A comparison of the boundary layer thickness in the CFD simulations, the experiments and the power law model for shear thinning fluids showed that the shear thinning effect in the pulp suspension flow seemed to have an impact on the boundary layer thickness as depicted in Figure 10. It is believed that the value of the yield stress has the greatest impact on the boundary layer thickness but the comparison of the boundary layers showed that the shear thinning effect might be necessary to take into account. The boundary layer thickness in the experiments reached a constant value which is more difficult to draw any conclusions about in the results obtained in the CFD simulations. It is probably the case in the CFD simulations as well since it is unrealistic that the boundary layer thickness decreased after a while which it seemed to do in Figure 9.
Figure 9: Comparison of CFD simulations and the Blasius solution.

Figure 10: Comparison of CFD simulation, experiment and the power law model.
4.2 Velocity profiles

The simulated velocity profiles were studied at \( x = 30 \text{ mm} \) and \( z = 18 \text{ mm} \) based on the boundary layer growth in both \( x \)-direction and \( z \)-direction over the plate. The velocity profile seemed to be close to fully developed at this point. The velocity distribution in the whole height of the pipe at this point is illustrated for different values of the yield stress in Figure 11. This illustration gives a good overview of the velocity profiles. The velocity profiles in the boundary layer is illustrated in Figure 12. The experimental velocity profiles were studied at \( x = 30 \text{ mm} \) since it was clearly fully developed there. The shape of the simulated velocity profiles were compared to the shape of the experimental velocity profiles in the boundary layer which is illustrated in Figure 13. The shape was very similar in the experimental and the simulated cases but it was on a smaller length scale in the experimental ones. The shape was not the same if just the boundary layers were compared since the boundary layer thickness was much larger in the CFD simulations. However, a higher value of the yield stress seemed to give a more consistent velocity profile.

Figure 11: Overview of the simulated velocity profiles in the whole height of the pipe.
Figure 12: Simulated velocity profiles in the boundary layer.

Figure 13: Experimental velocity profiles in the boundary layer obtained by Claesson et al (2013).
4.3 Pressure drop

The pressure drop was extracted from the CFD simulations along a line from $x = 20$ mm to $x = 200$ mm along the wall. The pressure drops for all the simulated cases is illustrated in Figure 14. The pressure drop seemed to be linearly dependent of the yield stress as well as of the velocity which was more difficult to draw any conclusions about in the results obtained by Claesson et al (2013). Their measured values of the pressure drops is illustrated in Figure 15. The measured values of the pressure drop was much lower than the ones obtained in the CFD simulations. However, the pressure drop increased with velocity and yield stress in both cases. A simulation of a lower yield stress, 10 Pa, resulted in a pressure drop that was in the order of the measured pressure drop in the experiments. This is an indication that the measured values of the yield stress actually were overestimated.

![Figure 14: Pressure drops in the CFD simulations.](image)
Figure 15: Pressure drops in the experiments by Claesson et al (2013).
5 Conclusions

The study of the boundary layer growth over the flat plate showed that the velocity profile was established faster in the experiments by Claesson et al (2013) compared to the CFD simulations. The boundary layer thickness was also thinner in the experiments which indicated that the shear thinning effect of the pulp suspension flow cannot be ignored. The velocity profiles were compared at points where it was fully developed and it was clear that a higher value of the yield stress gave a more similar velocity profile in the boundary layer when comparing the CFD simulations and the experimental results.

The difference in fluid properties between Newtonian fluids and pulp suspensions affected the boundary layer thickness. The boundary layer thickness of the pulp suspension flow was found to increase with an increase in velocity or a decrease in yield stress in contrast to the Newtonian boundary layer thickness described by the Blasius solution. The boundary layer thickness reached a constant value in the experiments which it seemed to do in the CFD simulations as well. According to Claesson et al (2013) was the constant boundary layer thickness a result of the yield stress properties and the two-phase effects in the pulp suspensions.

The near wall region is disrupted when the shear stress exceed the yield stress and the pulp suspensions start to flow. The boundary layer thickness will not be able to increase further when no more fibres are disrupted. A weaker fibre network is easier to disrupt which means that a flow at higher velocity disrupts the near wall region more than a flow at lower velocity. Therefore a thicker boundary layer thickness was observed when the velocity was increased which was the case both in the CFD simulations and in the experiments.

The pressure drops were also compared and they were much lower in the experiments than in the CFD simulations. The pressure drop increased with velocity and yield stress in both cases. A lower value of the yield stress gave a more realistic pressure drop. This is an indication that the measured values of the yield stress actually were overestimated in the experiments, as suspected. Another technique to measure the yield stress would be to prefer since the network strength might decrease strongly when the pulp suspension flow is established. However, this is a contradiction since lower values of the yield stress gave a worse agreement of the velocity profiles. Therefore, the wall treatment for the pulp suspension is suggested to be revised as well as the rheology of the fluid model.
References


