Application of Computational Fluid Dynamics in the development and optimization of stock preparation equipment

Andreas Gorton-Hülgerth, PhD – General Manager, Pilot Plant, Andritz AG, Graz, Austria
Jonathan Kerr – Executive Director, Paper Science & Engineering Foundation, Miami University

ABSTRACT

Computational fluid dynamics (CFD) simulation has become a more powerful tool in the development of pulp and paper equipment and processes. But simulating pulp flow is still very challenging as it is multiphase (water, fibers, ash, air, etc), the shape factor of the fibers is large, and often there is high and non-isotropic turbulence in certain areas of the equipment, making the modeling effort very complex and difficult, sometimes to the point of even putting the validity of the results in question. For example, in a single piece of equipment, the vessel might be quite large but the actual process area of interest could be very small (e.g., refining gap vs. total refiner). In such cases, a complete and overarching simulation is not reasonable and not a good industrial application of CFD technology.

But by carefully selecting the area of interest, selecting the right simulation models and making further simplifications and approximations, the calculations can give insights into the flow and function of certain equipment that cannot (at a reasonable cost) be gained by physical experiment. To find the right balance between needed simplification and over-simplification (which would provide meaningless calculation results) a constant exchange of information between the simulation and the real world is vital. The application of this approach on selected equipment in the stock preparation area results in general guidelines that help to speed up development at minimal comparative costs.

Finally, and most importantly, equipment designed using CFD achieves a higher efficiency when finally installed and operating, thus validating the cost and effectiveness of the modeling tool.

INTRODUCTION

Whenever we think about the production process of paper we have to deal with water, fibers, air and of course a lot of additional components like inorganic substances (ash), different chemicals etc. But the two main components are always fibers and water. The other substances have a smaller influence on the actual flow pattern. This is especially true for the stock preparation area. Therefore a solid understanding of the flow of a fiber suspension is crucial for the development and optimization of such equipment.

There are two main sources for gaining this knowledge. One is experimental data and measurement of certain flow parameters like velocity, pressure, consistency inside real equipment or models of the process units. The other one is theoretical calculation of these parameters. Theoretical calculations start with basic calculations like pressure losses in pipes and valves based on empirical or semi-empirical formulas and might end with Computational Fluid Dynamics (CFD) of the whole equipment.

As later discussed in more detail, the CFD calculation of the pulp suspension requires significant resources. Therefore it is important to define the additional information that should be gained by such a calculation before starting the process. One must also define the needed accuracy to get the maximum result at the minimum costs.

To verify the models used, a comparison of the calculated results to the measured values is mandatory. Without such constant comparisons, the calculation might give misguiding results and the optimization of the equipment could be going in the wrong direction.

THEORETICAL BACKGROUND

Conservation Equations

The basic equations for CFD are gained by setting up the conservation of mass and momentum over a small volume element.

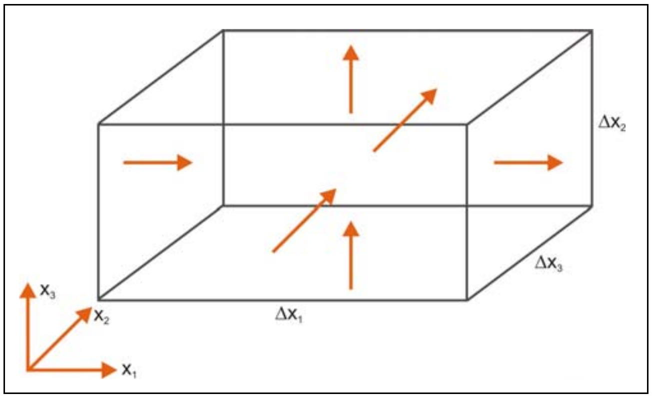

Figure 1: Flow through a rectangular volume

The conservation of mass (continuity equation) in differential form

$$\rho \frac{\partial U_1}{\partial x_1} + \rho \frac{\partial U_2}{\partial x_2} + \rho \frac{\partial U_3}{\partial x_3} = 0$$

or with Einstein notation

$$\rho \frac{\partial U_i}{\partial x_i} = 0$$

and momentum (Navier–Stokes equations)

$$\frac{\partial(\rho U_i)}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho U_i U_j\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{2}{3}\frac{\partial U_k}{\partial x_k}\delta_{ij}\right)\right] + \rho g_i$$

Convection terms pressure gradient diffusion gravitation

These are nonlinear partial differential equations that can only be solved numerically for basically every application of technical interest.

A direct numerical simulation would require an extremely small control volumes and resulting in extremely high computational times for any practical engineering applications. Therefore these conservation equations are time averaged, which means, for example, that a velocity is split into an average term and a fluctuating component ($U_i + u_i'$). As this fluctuating component is zero over long term only the so called Reynolds Stresses ($\overline{u_i' u_j'}$) remain in the Reynolds-averaged Navier–Stokes equations.

Turbulence Models

As these Reynolds Stresses introduce new unknowns, additional equations are needed to relate these to other variables. These are the so-called turbulence models.

The most common used industrial application is the k-ε Model. It introduces two additional variables the kinetic energy of turbulence, k, and the rate of dissipation of turbulence, ε. Using the Boussinesq Hypothesis it is assumed that the effective viscosity μ_{eff} can be expressed as the sum of the laminar viscosity μ and the so called eddy viscosity μ_t.

It is a semi-empirical model which is known for its robustness and stability. Therefore it is used as a starting point in most of the industrial CFD calculations. Only if the results of the calculation do not reflect the data close enough do more advanced models like RNG--ε Model or RSM Model have to be considered.

Non Newton Behavior of Pulp

Air and water are Newtonian fluids.

$$\tau = \mu \frac{du}{dy}$$

That means that there is a liner relationship between the shear stress and the velocity gradient. This viscosity depends on temperature and pressure, but not on the local strain.

Fiber suspension shows a totally different behavior.

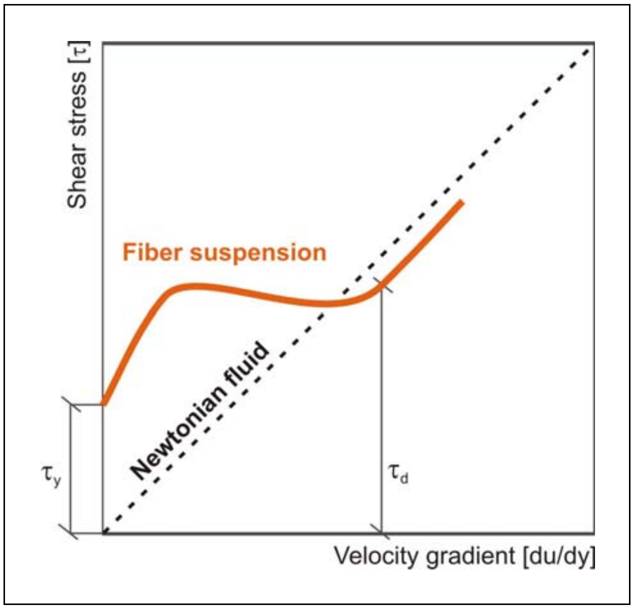

Figure 2: Non Newton behavior of pulp suspension

At very low velocity gradients, and especially at higher consistencies, the viscosity can be extremely high. But at higher shear rates there is a significant drop in the viscosity. This behavior is typical for shear thinning fluids.

The actual shape of the curve and the constants depend on the specific pulp used, consistency, temperature, pH etc. It is important to select the optimum consistency model for each application very carefully. It might be enough to handle the pulp suspension as water, but it might be that more advanced models are needed.

Multi phase models

Multi phase simulation is need as soon as there are several phases which are homogenized or separated in the process. There are different models available which are used depending on the actual target.

The volume of fluid model (VOF) assumes that there are separate immiscible phases. So every computation, a cell is either filled completely with one phase or there is an interface inside the cell. Such simulations are applied for flows with free surfaces e.g. the simulation of a pulper.

The Euler- Lagrangian model is tracking the flow of discrete particles inside the continuous phase. The maximum volume fraction of the discrete phase is about 10%. Depending on the complexity of the model momentum, mass and heat exchange are calculated. A typical application would be the rise of the air bubbles in a flotation cell.

The Euler-Euler models solve separate sets for momentum and continuity equations for every phase. So the concentration of every phase can be from 0% to 100% in a certain cell. This is the most complex model as the number of equations increases significantly with every phase.

PRACTICAL APPLICATION

Pulping

The target of repulping is to dissolve the dried pulp or paper with the smallest energy demand in the shortest time without having a negative impact on the fibers. For secondary fibers, the disintegration of any foreign material (like plastics or stickies) should be avoided as much as possible. Conventional pulping is done in an open vat with a special type of rotor which creates higher turbulence and shear forces than a mixing type agitator. The flow inside the pulper is defined by the rotor and the vat with its certain inserts.

Traditional development of a pulper rotor is done by manufacturing different prototypes and comparing their efficiency. But as the area around the rotor is a very harsh environment, the local flow around the rotor is difficult to measure. Also, the flow pattern inside the pulper is significantly different with water than with pulp, which makes every measurement that much more difficult.

CFD is therefore a useful tool to get an insight into the area close to the rotor and define additional characteristics like the pumping curve of the rotor.

Models used for CFD calculations:

- k-ε Model
- Volume of fluid model (VOF)
- Constant consistency
- Steady state solution

Use of the above modeling techniques means that the dissolving part of the pulping progress itself is not simulated, but only the movement of the “dissolved” pulp after it has reached stable conditions.

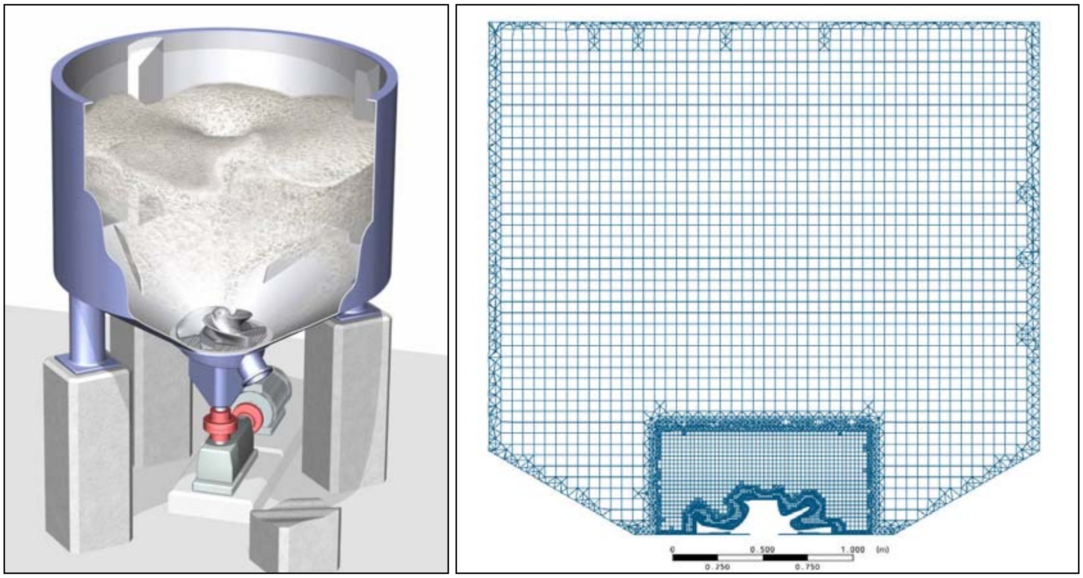

Figure 3: Model and grid of the pulper

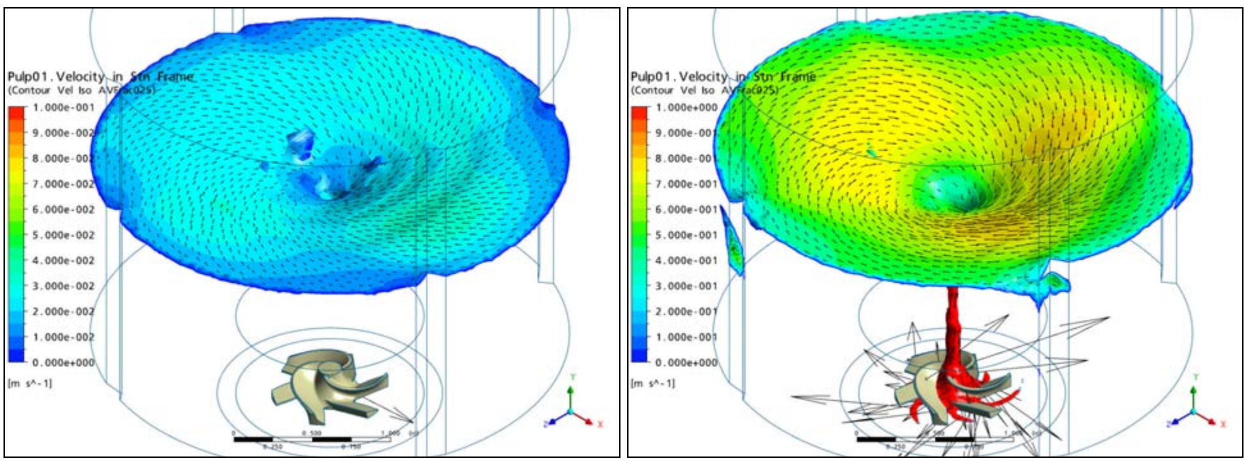

Figure 4: Influence of Rotor velocity on surface shape and speed

The first step when performing calculations is to compare the results to real data to check the accuracy of the simulation. Figure 4 shows the speed of the pulp surface at low (left) and high (right) rotor speeds, which are then compared to measured speeds in a real pulper. (The high speed model clearly shows that at such speeds, the rotor pulls air down towards its center, thus decreasing the efficiency of the rotor.)

After evaluation of the model with different similar basic tests, sufficient confidence in the quality of the simulation models can be gained so that the models are considered valid for the chosen conditions.

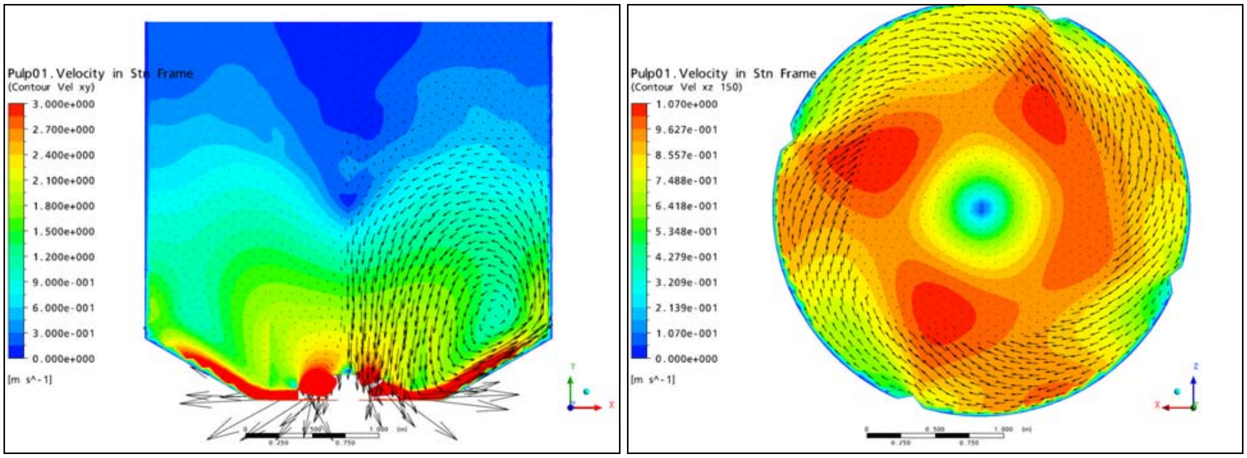

Figure 5: Velocity in a vertical plane through the rotor and in a horizontal plane 1.5 m above the rotor

Figure 5 shows the velocity in a vertical plane through the center of the pulper and a horizontal plane 1.5 m above the rotor. Because of the VOF model used, it is not possible to differentiate between the pulp phase and the air phase in this figure, but it can be noted that the upper and blue area on the left side is part of the air phase.

These velocity figures allow us to optimize the shape of the rotor and the vat simultaneously. It can be checked that there are no dead zones with low velocity which could lead to uneven pulping. At the same time extreme velocity peaks which just cost energy can be avoided.

Pulping is most efficiently done by shear strain inside the fluid. To be efficient, the strain has to be above a certain level to be effective. The efficacy of the strain applied is checked by viewing the values in different planes inside the pulper.

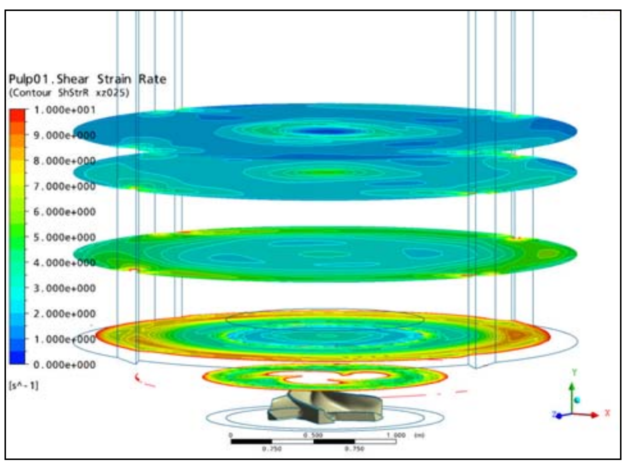

Figure 6: Shear strain inside the pulper vat

The active pulping volume can be maximized by assuring that there is sufficient shear strain at the walls of the pulper vat. Note that the design value of the shear strain must first be defined experimentally.

An additional feature of CFD is the option to define certain volumes or surfaces inside the calculation area. An example for such surfaces is shown below.

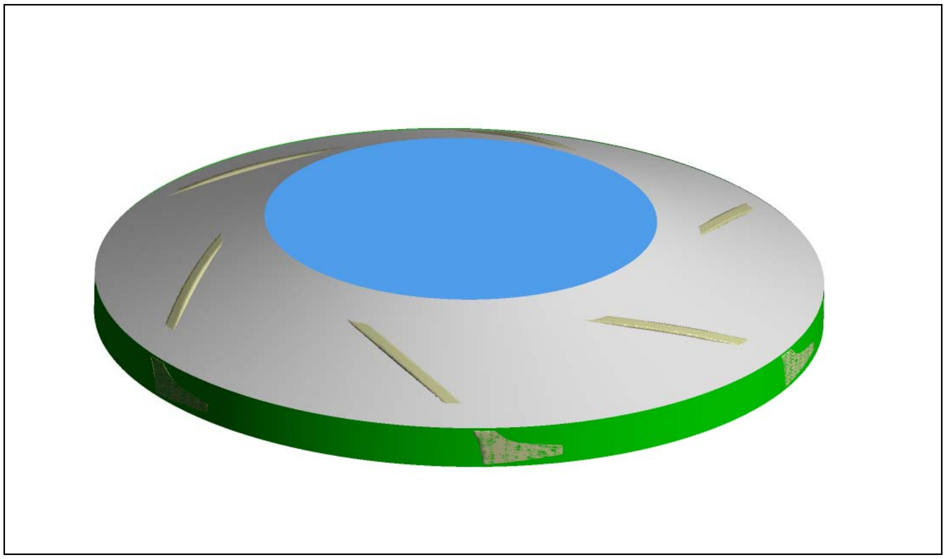

Figure 7: Control surfaces around the rotor

The blue zone defines the inflow area where the rotor is pulling down the pulp suspension. The grey area is the zone of intensive mixing. The green zone is the outflow area where the pumping work of the rotor is done.

The flow through such surfaces can be easily calculated by the CFD program. By simultaneously calculating the absorbed power of the rotor, the most efficient speed of the rotor can be determined, and the pumping curves defining the relationships between the speed, absorbed power, pumped volume, etc. can also be defined.

Such curves allow a quantitative comparison of different rotor designs. This is especially beneficial for optimization studies where just relying on visual observations of the results might be misleading.

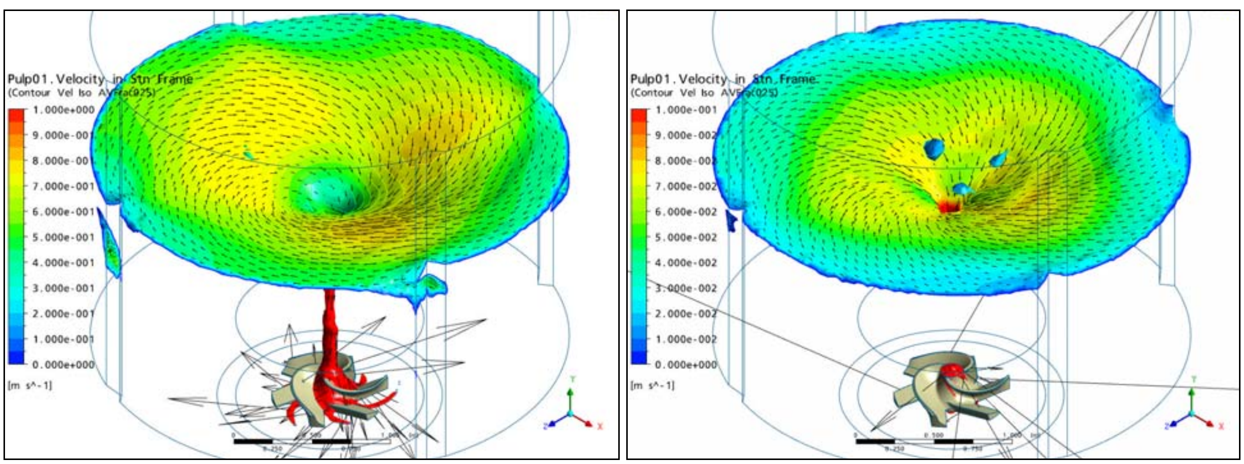

Figure 8: Influence of pulp consistency on surface shape and speed

Figure 8 is demonstrating the influence of the pulp consistency on the simulation results. The left picture is at low consistency the right picture at higher consistency. When increasing the consistency, a slowing of the surface motion can be observed. At the same time the extension of the vortex in the middle is also decreased significantly. The model shows an extremely good correlation to the flow pattern of a real pulper, thus validating the assumptions of the viscosity model

Disperser

A disperser is used in recycling application for disintegration of dirt and stickies. In general, this process is done at a consistency of about 30%. In some applications a low consistency (LC) outlet is used where water is added after the dispersing zone to allow transportation of the pulp using pressure instead of transfer screws.

When adding the water to an LC discharge, it is important not to get water into the dispersing zone itself, so as not to destroy the efficiency of the dirt and sticky reduction. At the same time, a homogenous and even mixing is needed in the discharge zone to allow for the stable operation of the following equipment. And finally, if there is storage after the disperser, the consistency after disperser should be in the medium consistency (MC) range to minimize the required storage volume.

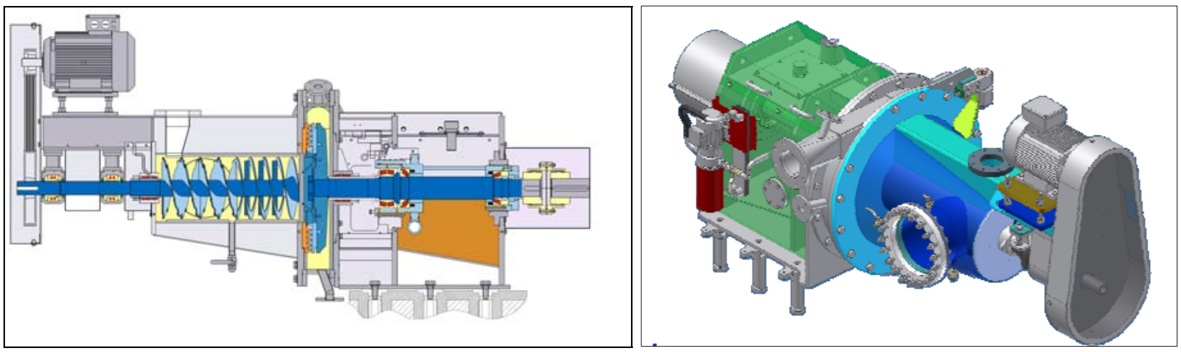

Figure 9: Disperser with LC discharge

The CFD calculation in the disperser has some special challenges that would make a simulation of the complete machine extremely difficult. The feed is done at HC (about 30%) and steam is added into the feeding screw to heat up to about 360 K. After that, the pulp is dispersed in a small gap (around 1 mm). The circumferential speed of the disc is up to 60 m/s which creates some very high velocity gradients. In the outer zone, pure water is added so the consistency inside the machine ranges from 30%, which is a granular flow, to 0% of the pure water.

For the purposes of the simulation, it was decided to simulate just the outlet zone of the housing. The disperser plates are approximated by flat discs (neglecting their tooth design), and the outlet pipe was extended to achieve stable outflow conditions.

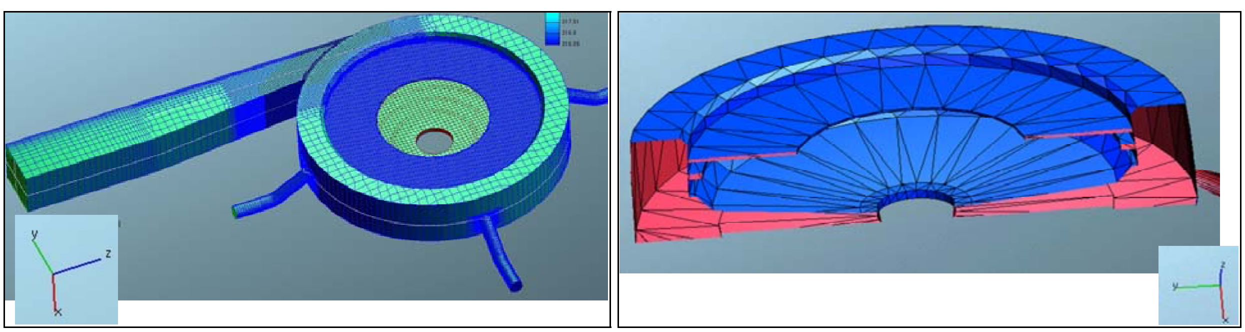

Figure 10: Grid of the disperser housing

The influence of the correct consistency model was also studied in this instance. In Case 1 the pulp phase is approximated as water, which is of course wrong, but this assumption is faster and easier to calculate. In Case 2 an Euler-Euler approach was used for the water and pulp phase, and a viscosity model was used to reflect the change in viscosity caused by the consistency range. The viscosity model reaches its limit at the modeled consistency, but having Case 1 as the other extreme, the sensitivity of the solution could be studied.

Models used for CFD calculations:

- k-ε Model
- Case 1: single phase with scalar marking of the pulp
- Case 2: Euler-Euler two phase model with variant consistency
- Steady state solution

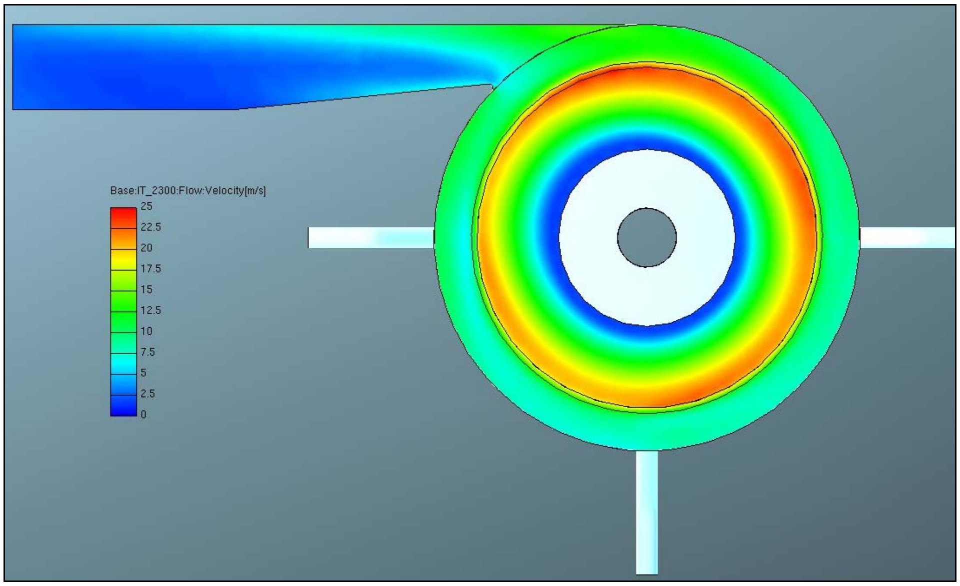

Figure 11: Velocity in the plane of the disperser gap

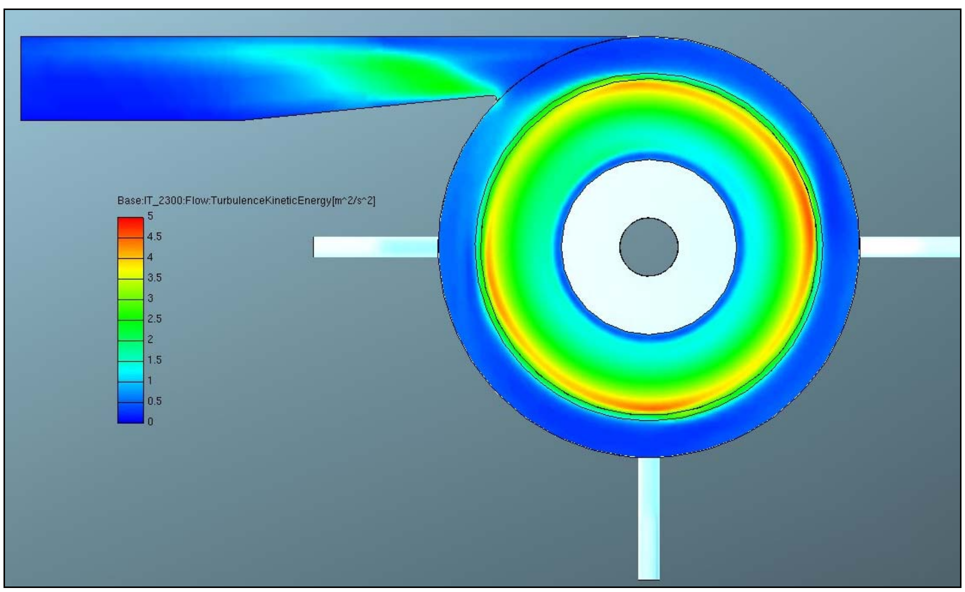

Figure 12: TKE in the plane of the disperser gap

Figure 12 shows the velocity and Figure 12 shows the turbulent kinetic energy in the plane of the disperser gap. The values inside the gap are known to be an approximation as teeth inside the gap have been neglected and modeled by a flat disc. Nevertheless, pulp reaches a velocity of about 25 m/s inside the gap, but is then very soon decelerates to a velocity of about 10 m/s in the housing. This causes a stable homogenization of the pulp with the water. There is practically no influence of the water inlets on either the velocity or TKE, as can be seen in the illustration of the plane of the disperser gap. At the beginning of the outlet pipe there is an area of high turbulent kinetic energy which causes additional mixing and homogenization of the pulp.

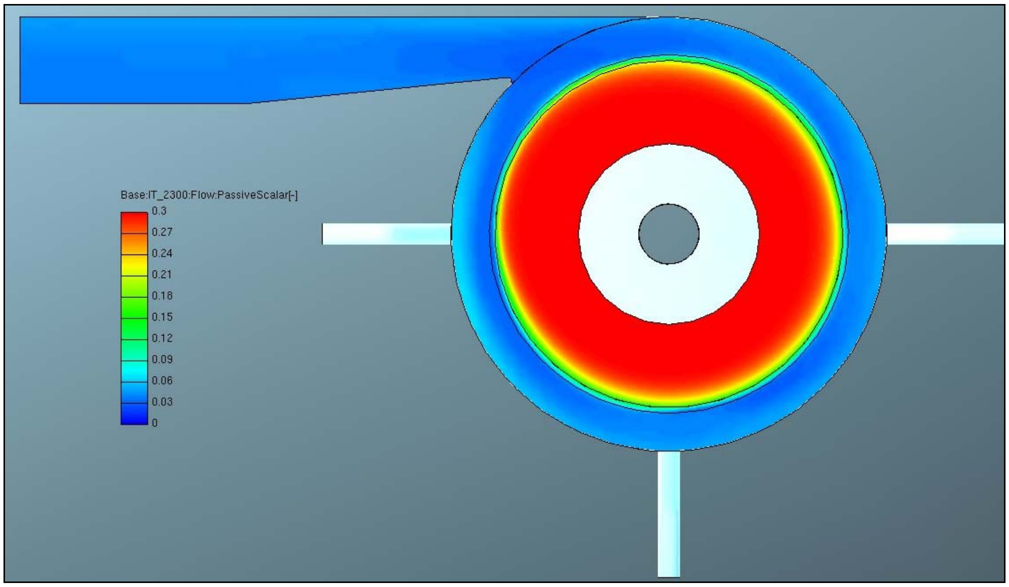

Figure 13: Mass fraction of pulp in the plane of the disperser gap

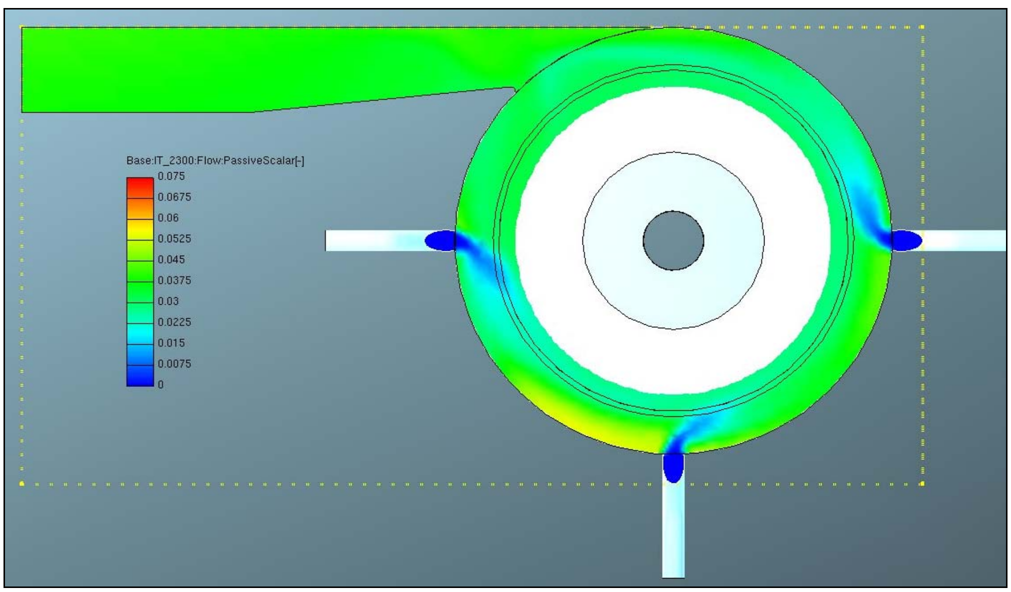

Figure 14: Mass fraction of pulp in the plane at the water inlets

Figure 14 shows the mass fraction (consistency) of the pulp at the plane of the disperser gap and Figure 14 shows consistency at the water inlets. Notice that the maximum consistency (red color) of Fig 13 is 30% and 7.5% of Fig 14. Also in Fig 14, the three water inlets cause significant variance in consistency around the rotor. But at the most important plane of the disperser gap, no influence of the locations of the water inlets can be seen. There is also a very sharp border between the circumferential housing area and the disperser gap, so no negative influence of the water addition on dispersing efficiency can be expected.

All the results presented above were done with the setup of Case 1, which was preferred because it is numerically much simpler and therefore faster to solve, however the final geometry was also solved for Case 2.

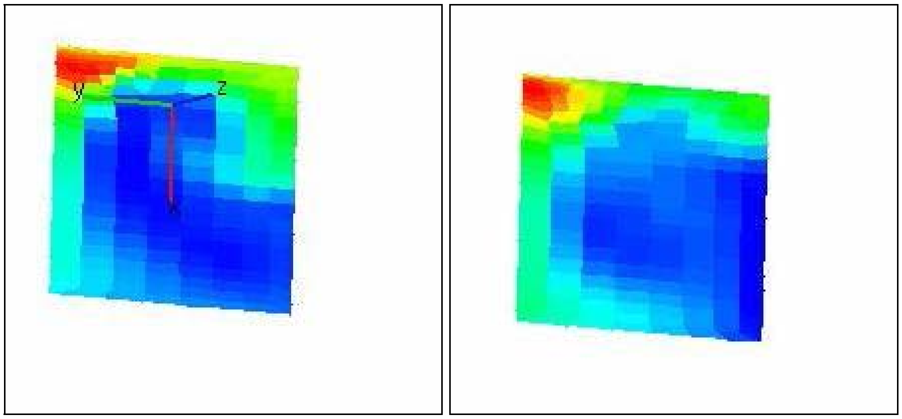

Figure 15: Comparison of consistency variations for case 1 (left) and case 2 (right) in the outlet pipe

Figure 15 shows the consistency variations in the outlet pipe for both cases. The minimum consistency is about 3% the maximum 5%. There are only minor differences between these two models which at first was very surprising. But the high velocity of the rotor creates extremely high velocity gradients which result in an apparent viscosity of the pulp phase that is very similar to water, which confirms that our initial approximation was accurate after all.

SUMMARY

The above examples of the application of CFD on stock preparation machines represent some basic ideas of the information that can be gained. Similar examples can be found on almost every piece of equipment in this area of the mill.

To achieve the maximum results from the calculations, some important questions have to be answered before and during the modeling process:

- What additional information should be obtained by CFD?
- Is it needed to simulate the whole machine or just certain area of interest?
- How much simplification is possible without disturbing the results?
- Do we have models to simulate the process fully?
- How to compare the calculated results to experimental data to validate the approach?

Of course this is an iterative process, which becomes especially important as soon as there are big deviations between the results of the calculations and the experiments, indicating that the models and assumptions have to be improved.

As shown in the examples above, the consistency of the pulp has to be considered in the pulper simulation to get the correct results. But at the disperser, where the consistency is changing much more and the first idea was that the influence is even higher, the simulation did not show any significant changes in the results (depending on the viscosity model used). Such comparisons help to simplify the modeling as much as possible and save computation time to a great extent.

Based on the CFD calculations, a much better understanding of the process inside the equipment can be obtained and studies of the effects of certain design parameters on the flow can easily be done. Finally, and most importantly, CFD modeling leads to equipment designs that achieve a higher efficiency when finally installed and operating, and the development work is done with minimum development time and costs.

REFERENCES

1. E. M. Marshall and A. Bakker, “Computational Fluid Mixing”, Fluent Inc., 2002
2. J. Schurz and T. Tyralski, “Rheologie von Zellstoff-Suspensionen”, Das Papier 10A (1987) 19-26 in German
3. K. Gustavsson and J. Oppelstrup, “Consolidation of concentrated suspensions - numerical simulations using a two-phase fluid model”, Computing and Visualization in Science 3 (2000) 39-45