SimSpire's practical and cost-effective approach to CFD

Thomas Sprich of SimSpire talks about the use of computational flow dynamics (CFD) for improving process-mixer, pump and fan design: the problems that can be solved; how CFD works with physical testing; and how to simplify the models to get best-value results.

Computational fluid dynamics (CFD) is a way of simulating fluid flow for process plants, HVAC circuits, pumps, fans, mixers, water treatment systems, pipelines and an endless list of equipment designs that involve flow of any kind. "CFD involves the numerical modelling of fluid flow to better understand its behaviour, ultimately with the intention of validating designs or to improve upon existing designs," begins Thomas Sprich of SimSpire, a CFD service specialising in process equipment, mixer, pump and fan simulations.

"In CFD we use finite volumes to create models of how fluids will flow and how that flow will affect the objects around or suspended in those fluids," Sprich says. CFD software, he continues, typically solves the Navier–Stokes equations, which numerically describe fluid motion for a given geometry.

"CFD solvers must first divide the fluid domain into millions of sub-volumes called mesh cells. The Navier-Stokes equations are solved for each of these mesh cells ensuring that continuity is maintained, boundary conditions are adhered to and any other more complex models are applied.

"When a solution is found, pressure, temperature, velocity and turbulence information at each cell location is available for further analysis. In addition, if solving a transient problem, this information is available for each time step as well," explains Sprich.

"Simply put, CFD breaks a volume down into a network of sub-volumes and then solves momentum and continuity equations for each mesh cell.

"In the fluid mechanics world, turbulence is a significant challenge, which is hard to solve because the scale of turbulence can go from the whole contained volume all the way down to the micro scale. Resolving the mesh at the smaller scale while still obtaining results in a reasonable time is still impossible, so we have to resort to some simplifying assumptions and empirical models to extract useful information about turbulence for real applications," Sprich points out.

Called SimSpire, Sprich's company was established to meet the needs of equipment suppliers that cannot justify the expense of a CFD 'seat' inside their engineering department. To implement CFD successfully, equipment designers not only need to purchase a CFD package, they also need a specialist engineer with a high-end computer and the skills to model and run simulations and extract meaningful results.

"Most CFD packages, while being expensive, are able to solve a wide range of problems such as incompressible flow, supersonic flow, heat transfer or multiphase flows. Engineers designing mixers, pumps, fans and minerals processing solutions, however, probably don't need these advanced CFD features, despite having paid for them," argues Sprich.

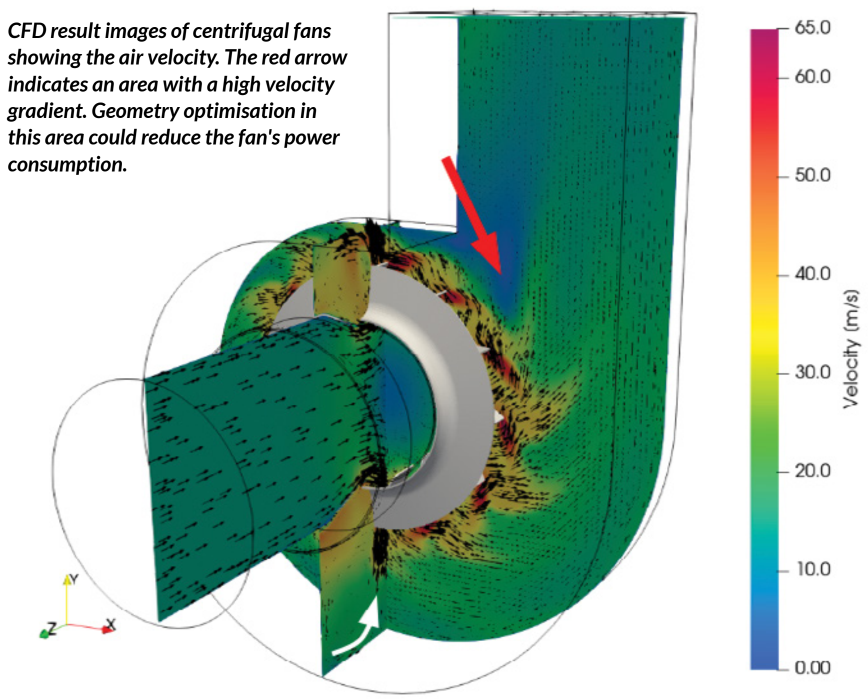
SimSpire offers a far more cost-effective way of incorporating CFD simulations into the designs of processing equipment. "We offer a CFD service to those wanting to improve equipment performance without having to employ new people or invest in high-end technologies. If a company needs to improve the efficiency of specific mixer on a specific mine, we can upload the geometry and enter the liquid properties and the mixer speed. The mesh is then automatically set-up and a simple simulation is run.

"After checking and analysing the results, we generate a report for the equipment engineers," he notes. "This process enables the richness of CFD to be incorporated as a low-cost value-add. Multiple configurations can then be explored for a deeper understanding of the most influential design components that affect performance and efficiency.

"We tackle standard problems where it makes engineering and economic sense to run CFDs, but doesn't make sense to employ a fulltime CFD specialist," he adds.

Key to this approach is to keep the CFD analysis as simple as possible. Complicated simulations can become difficult to understand and believe, because they tend to include underlying assumptions that may not be applicable. But if you start with the simple case and systematically build up a model that makes practical sense every step of the way, then confidence grows.

"This also helps in managing computational costs, which are dependent on the number of mesh volumes. Sometimes, for example, a simulation with a 2D mesh can produce very useful results – and these cases can be run on a laptop in 15 minutes," he says.

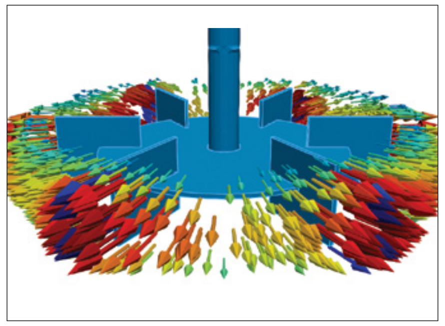


CFD result images of centrifugal fans showing the air velocity. The red arrow indicates an area with a high velocity gradient. Geometry optimisation in this area could reduce the fan's power consumption.

Citing some examples, Sprich says that, while mixers are quite central for many process applications, the technology is seen as well-established and it isn't changing fast. "Customers will specify a required input power per unit, which they expect to see, even if a simulation suggests that a modified impeller can deliver the same duty but at a lower power," he notes.

The role of CFD is to convince process equipment manufacturers that it is possible to achieve better mixing efficiency while also reducing the power draw, for example, and South Africa's current power constraints are tending to push engineers in this direction.

First and foremost, the engineered solution must meet the duty required. "For mixers in mining applications, there typically needs to be enough turbulence in the tank to suitably mix the entire volume within the residence time. Minerals particles must be equally suspended in all areas of the tank for the full reaction time. This relates back to flow velocities, which have to be sufficient to overcome particle settling rates throughout the tank. From an energy optimisation point of view, CFD can help to reduce the input power needed while still fully satisfying minimum velocity criteria," Sprich tells *MechChem Africa*.



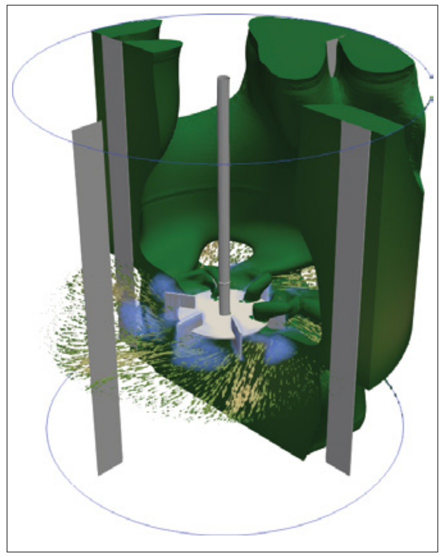
A CFD of a radial flow impeller. The arrows indicate the velocity of the water off the impellers. Highest velocities occur at the blade tips with the lowest velocity between two blades.

"This minimum velocity is also likely to be furthest away from the impeller, with all of the absorbed power being introduced at the impeller. By using two or three impellers to better distribute power through the tank, it is often possible to reduce impeller speeds while achieving better particle velocity distribution – with less input power," he points out.

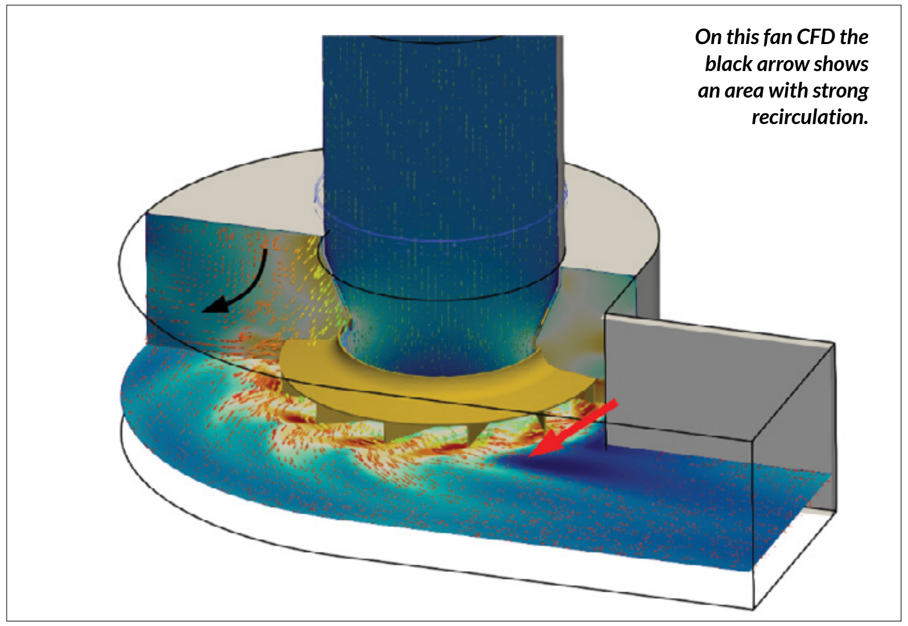
On the mechanical side, he says that CFD results provide the pressure distribution across the impeller blades of a mixer, pump or fan and, by understanding that better, engineers are able to improve their blade designs – adjusting blade profiles to make them stiffer and/or stronger where needed or to make them lighter and less expensive. "A thinner impeller is also easier to fabricate, handle, package, deliver and install, all of which adds to the value of using CFD to identify opportunities," says Sprich.

Tank design is another area of opportunity. "The tank design of minerals processing equipment is the orphaned child that no one optimises. Tanks are often cylindrical with flat bottoms.

"But vessels in the chemical process industry have dished-ends. As soon as you see the flow patterns of CFD simulation, it becomes obvious that square corners are bad for flow. While incorporating an expensive dished end on the bottom of a process tank is clearly unfeasible, it is quite easy to add a simple chamfer around the bottom edge. I estimate that a mixer tank with a simple chamfered floor could deliver 10 to 15% better mixing efficiency compared to using a flat-bottomed tank – and in terms of costs to client, this is virtually free," he adds.



A CFD of a radial flow mixer showing how baffles can assist in better generating mixing at the top of the tank.



On this fan CFD the black arrow shows an area with strong recirculation.

As with any simulation, however, Sprich notes the importance of scrutinising the results of every CFD analysis, especially for results that are exactly in line with expectations. "It is important to constantly interrogate all results to keep trying to find more evidence to either prove it or disprove simulation.

"Real life validation is also a must. On process mixers, test work can be done, but usually at laboratory scale rather than at actual system size – and scaling the results can be tricky. Generally, though, it is possible and necessary to validate CFD results by measuring what happens in reality, via a test or on an installed design. Then, once confidence has been established, the use of CFD can be expanded into areas where test work is more difficult," he advises.

Due to cloud-based services, the computational costs of CFD are coming down and there are now a number of open source software alternatives that are properly investigated and scrutinised. "This is making CFD simulations much more accessible to local equipment designers seeking to systematically improve the performance, efficiency and cost effectiveness of the products they supply into industry," concludes Sprich.

simspire.com