IN THE MIX

By Dave Grutzmacher

April 2017 – Issue 2

Computational Fluid Dynamics Series – Part 2 of 4

Applying 3D Computational Fluid Dynamics Techniques in Tank Agitator Design

We hope you found Part 1 of our blog series on Computational Fluid Dynamics (CFD) helpful. In it, we discussed how computational fluid dynamics predicts flow patterns in both simple and complex geometries including mixing applications. The second part of our blog series on Computational Fluid Dynamics looks at the use of 3D CFD techniques in tank agitator design.

Significant Advances Made in Design and Use of Simulations for Stirred Tanks

One of the most common chemical plant operations involves agitation in a stirred tank. This, however, presents one of greatest challenges in the use of computational fluid simulations. Significant advances in the design and use of simulations for stirred tanks have overcome many of these challenges. Today's 3D CFD techniques can be used to optimize the design of impellers, mechanical components and tank internals (i.e., baffles, coils). This allows predicting process variables such as blend times or suspended solids cloud height.

While 3D Computational Fluid Dynamics has become an important tool in tank agitator design, it is most successfully applied by experienced engineers with in depth knowledge of applicable software, analysis tools, and mixing technology. Significant computing power is also required.

3D Computational Fluid Dynamics Techniques

2D computational fluid dynamics modeling, which had been commonly used in tank agitator design, required approximations to be made for mixing components that are not independent of the angular dimensions such as impellers, baffles and other internals. This resulted in an approximation of the mixing flow pattern in the vertical plane.

Today, advancements in both modeling techniques and computational power have allowed routine use of 3D computational modeling. 3D modeling takes into account complete impeller geometry and its rotation in the vessel. This has made it possible to analyze more complex impeller geometries with greater accuracy.

3D Computational Fluid Dynamics can be used to show mixing patterns from complex impeller geometries as illustrated below. A commonly used 3D CFD analyses is a Velocity Contours Plot which shows the distribution of fluid velocity magnitude in the vessel. This is used in conjunction with a Velocity Vector plot which provides a visual representation of fluid flow circulation, flow pattern and range of fluid velocities in the vessel.

The 3D CFD analysis shown is for ProQuip's Doubly Pitched Axial Turbine Impeller design in a non-baffled tank. This impeller is capable of providing top to bottom circulation of high viscosity fluids in a non-baffled vessel. Figure 1 is a Velocity Contours plot for the Doubly Pitched Axial impeller. Figure 2 is the Velocity Vector plot for this impeller.

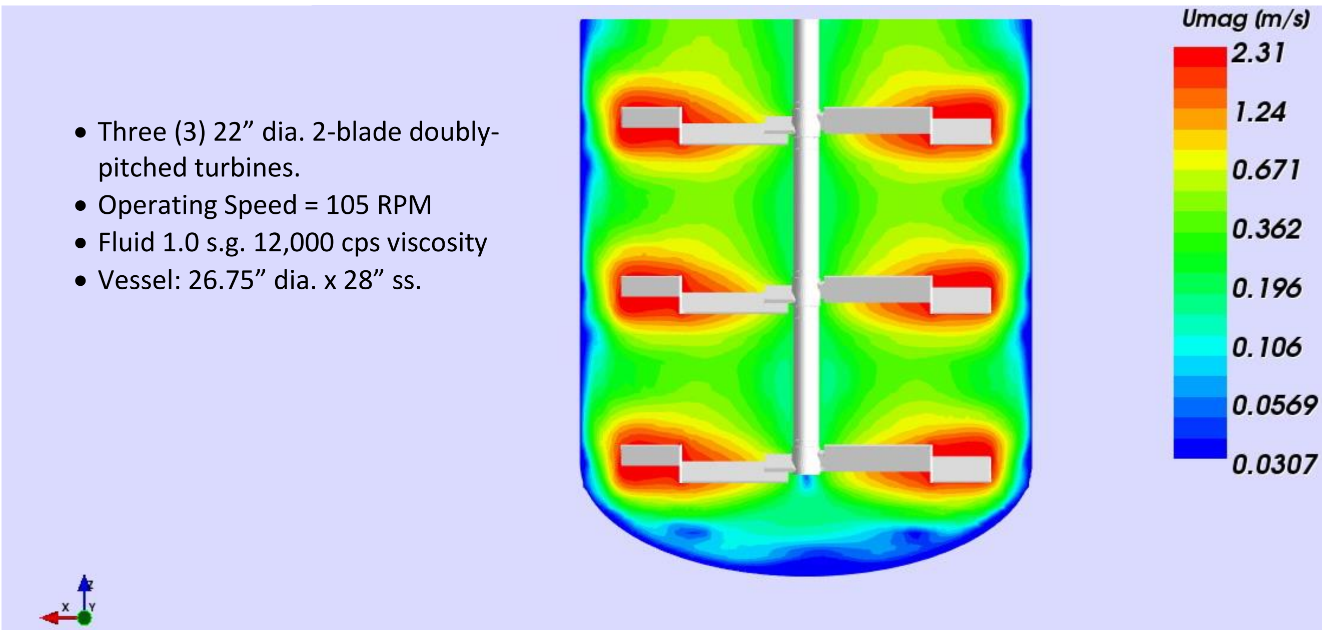

Figure 1. Velocity Contours Plot with a ProQuip Doubly Pitched Axial impeller.

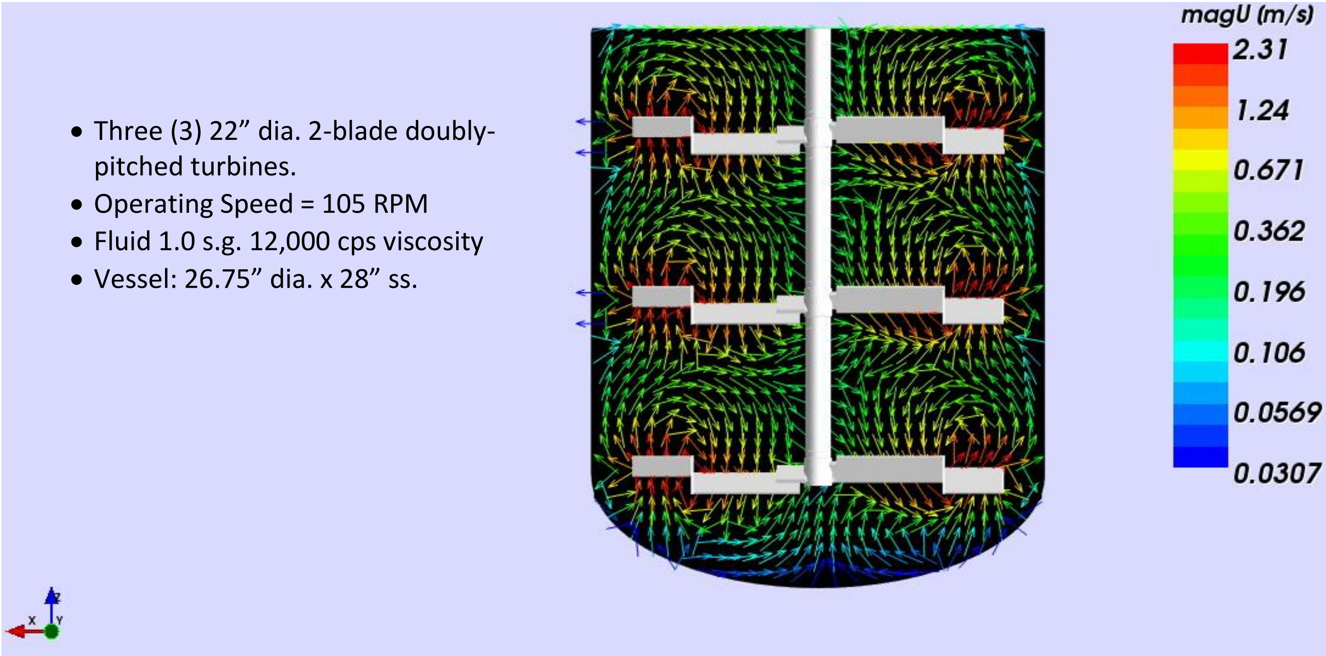

Figure 2. Velocity Vectors Plot with a ProQuip Doubly Pitched Axial impeller.

For More Information on 3D Computational Fluid Dynamics Techniques

For more information about how 3D computational fluid dynamics can be utilized in your tank agitator design, look for Part 3 of our series on CFD, email ProQuip at applications@proquipinc.com or call us at 330-468-1850.