Simcenter STAR-CCM+

Enabling MSI to analyze complex pump hydraulics

Mechanical Solutions Inc. (MSI) has successfully incorporated Simcenter STAR-CCM+® software into its process chain as a design and troubleshooting tool. The advanced, accurate flow solver in Simcenter STAR-CCM+ offers MSI a streamlined, cost-efficient engineering process to solve complex problems in hydraulic turbomachinery.

A white paper issued by: Siemens PLM Software

Contents

Executive summary

Turbomachinery designers are turning to advanced design methodologies using computer-aided design/engineering/manufacturing (CAD/CAE/CAM) tools for virtual prototyping, optimization and troubleshooting. Such simulation tools enable designers to accurately predict performance earlier in the design cycle, analyze multiple designs, reduce reliance on multiple physical prototypes and expensive testing, optimize design for maximum performance and shorten design time and cost.

Mechanical Solutions Inc. (MSI), headquartered in Whippany, New Jersey, specializes in fluid machinery design, fluid dynamic analysis and mechanical design and analysis. In the energy industry, MSI focuses on the analysis, testing and troubleshooting of all kinds of rotating, reciprocating and turbomachinery, using sophisticated analysis and testing tools. This white paper shows how Simcenter STAR-CCM+, an industry-leading simulation package from Siemens PLM Software, enables MSI to tackle design challenges, and design better performing pumps faster. Dr. Ed Bennett of MSI summarized his team's experience with Simcenter STAR-CCM+ by saying, "Simcenter STAR-CCM+ has all the necessary features to analyze the most complex flows in hydraulic turbomachinery."

Simulating water pumps with Simcenter STAR-CCM+

Water pumps are widely used in fossil fuel, nuclear and hydropower plants as well as chemical processing, automotive cooling, oil and gas and waste management. Major design challenges of water pumps include the unsteadiness of the flow, cavitation and rotating machinery. Simcenter STAR-CCM+ offers features that can be used to tackle these difficult physics while offering a streamlined, robust numerical design process. Simcenter STAR-CCM+ allows you to create an efficient workflow for generating geometry, meshing, solving, postprocessing and optimization.

Features like the unsteady flow solver, unsteady cavitation model, rigid body motion (RBM) for rotating domains, automated unstructured meshing for complex geometry and parallel processing capability offer the ability to explore multiple water pump designs in an economical manner. Using Simcenter STAR-CCM+ allows designers to easily predict pump performance at design and off-design points and avoid the damaging effects of cavitation and erosion.

Analyzing a double suction pump

MSI was tasked by a customer to analyze a double suction pump design to verify whether the pump could meet its performance requirements. The pump was a complex centrifugal design for enhanced cavitation performance. It included an inlet from the top with a splitter down the middle to mitigate radial loads and featured identical impellers on either side leading up to a volute. The flow entering the pump was equally split across the two impeller sections. MSI used Simcenter STAR-CCM+ to simulate the complex physics of the pump, including transient flow through 360 degrees, rotating geometry and unsteady cavitation. The flow path was cut in half due to symmetry for simulation, and the inlet, impeller and volute were modeled. The automated polyhedral meshing capability of Simcenter STAR-CCM+ was used to discretize the flow domain. A mesh for a single vane passage was created and cyclically patterned and fused to create the complete impeller mesh and ensure mesh uniformity. Prism layer cells were created automatically on surfaces to resolve the boundary layer; the final mesh had 5 million volume cells. Figure 1 shows the geometry and mesh of the flow path.

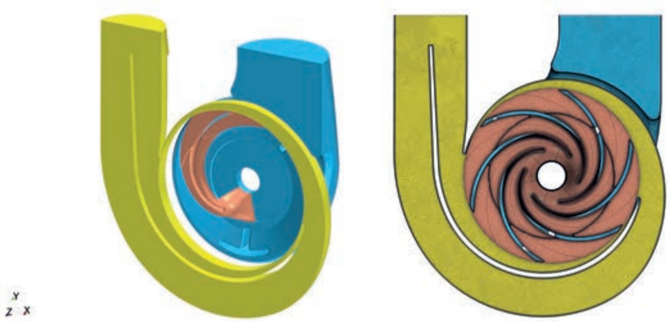


Figure 1: Flow-path geometry of the pump (left) and computational mesh in Simcenter STAR-CCM+ (right) along the center plane.

The segregated flow solver in Simcenter STAR-CCM+ was used with a second order convection scheme and the shear stress transport (SST) turbulence model. The multiphase volume of fluid (VOF) model was activated to capture the interface between water and water vapor. The Rayleigh-Plesset cavitation model was selected for simulating cavitation. Total pressure conditions were given at the inlet via a reference point and mass flow was specified at both the inlet and exit. The rotating impeller was surrounded by an interface enclosing it and the domain was given a rotating speed matching the impeller speed using RBM. The unsteady time step and total time were chosen to simulate one complete rotation of the impeller; 20 inner iterations per time step were used. The simulation was run for several complete rotations until the residual monitors were settled and variable monitors for pressure, torque and mass flow showed cyclical behavior.

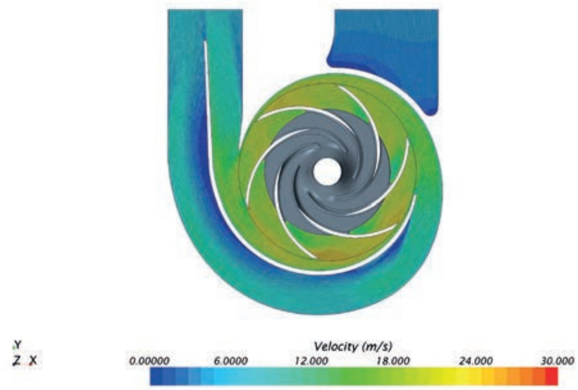


Figure 2: Velocity contours shown as line integral convolution at center plane.

The velocity contours (figure 2) showed stationary flow around the impeller, a nice uniform volute flow and recirculation near the splitter. Some recirculation was noticed near the inlet and volute due to the limited axial size of the pump. The nonuniformity of flow was deemed undesirable but had to be accepted because increasing axial space for more uniform flow was not possible with the limited axial space available. Figure 3 shows contours of vapor fraction on the impeller with blue regions representing water and red representing water vapor. It can be seen that at a high inlet pressure (175 kilopascals), there is very minimal cavitation and as the inlet total pressure decreases, cavitation on the impeller increases.

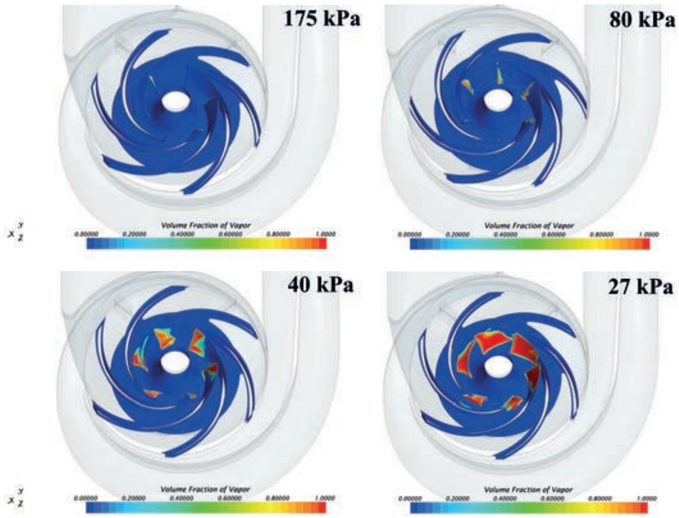
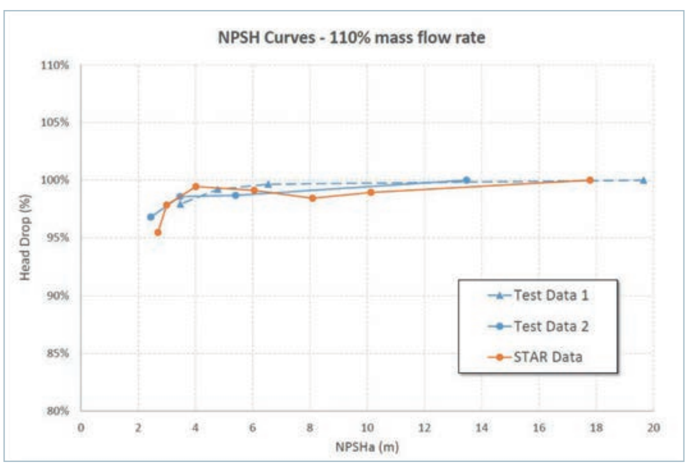
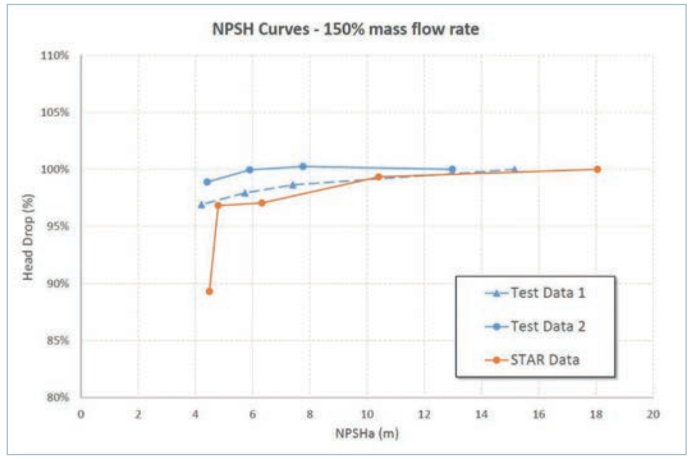


Figure 3: Vapor fraction contours on the impeller showing cavitation at various inlet total pressure values.

Figure 4 shows the comparison of net positive suction head (NPSH) measured by the manufacturer in a test rig and computed by Simcenter STAR-CCM+ at different mass flow rates. Simcenter STAR-CCM+ accurately captured the breakdown of pump performance at lower NPSH and lower inlet pressure, and the results compare favorably with the experimental data. Using Simcenter STAR-CCM+, MSI numerically created the NPSH breakdown curve for the pump at 70 percent, 110 percent and 150 percent mass flow rates. This allowed the customer to make clear design and certification decisions, including cavitation performance for the pump based on the Simcenter STAR-CCM+ simulations.





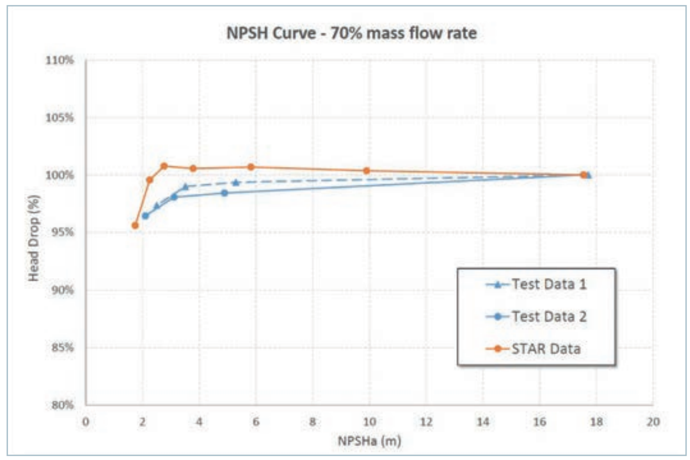


Figure 4: Cavitation breakdown curve comparison between Simcenter STAR-CCM+ and test data showing NPSH versus head drop for three different flow rates.

Simulation of a vertical flood control pump

MSI was tasked with analyzing the pump design of a large axial pump for the city of New Orleans for resonance and vibration issues resulting from an unsteady, forced response that could be expected during severe weather conditions. These new vertical flood control pumps were intended to provide durable performance and remove excess water from the city during flooding.

The pump had a large inlet, an impeller with a wheel diameter of over one meter, a vane diffuser, an internal shaft, a discharge vane propped by guide vanes and an exit siphon. The complex geometry of the pump was imported into Simcenter STAR-CCM+ and discretized using the automated polyhedral and prism layer meshing capability. The final computational mesh had 4 million volumetric cells and accurately captured the complex geometry with multiple channels and flow passages. Figure 5 shows the complex geometry and computational mesh of the flow path.

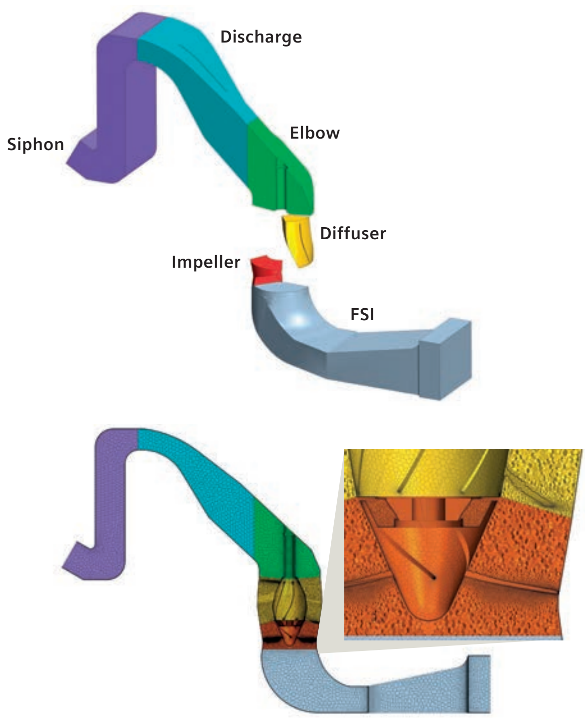


Figure 5: Vertical pump geometry (left) and computational mesh along the center plane (right).

The segregated flow solver in Simcenter STAR-CCM+ was chosen with the second order convection scheme and realizable k-epsilon turbulence model. Water was chosen as the working fluid at standard temperature and pressure. A mass flow boundary condition was specified at the inlet, static pressure at the exit and rotating speed on the domain enclosing the impeller was specified as well. The transient simulation was run to complete one full impeller rotation with 20 inner iterations per time step. The simulation was run for several complete rotations until the residual monitors were settled and variable monitors for pressure, torque and mass flow showed cyclical behavior.

Figure 6 shows contours of velocity at the centerline plane of the pump. The image shows vortical flow through the discharge region created by the shaft. Simcenter STAR-CCM+ has been well validated at MSI and as such, pressure traces from the simulations were used for frequency analysis with National Instruments LabVIEW software instead of experimental values. The pressure at every interface in the domain was processed in frequency domain in LabVIEW using fast Fourier transform (FFT) for spectral analysis. The frequencies of oscillation in the flow path were captured in this analysis, including vane pass frequency, impeller frequency and two times vane pass (figure 7). The captured frequencies were analyzed in a modal analysis software and no vibration/resonance issues were found in the pump design.

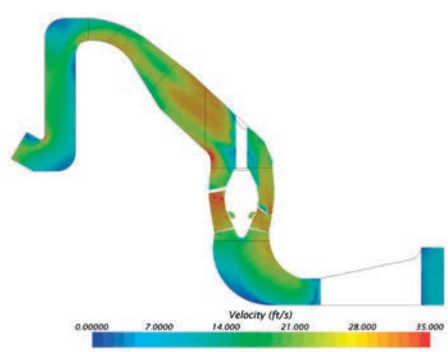


Figure 6: Velocity contours at center plane shown as a line integral convolution plot.

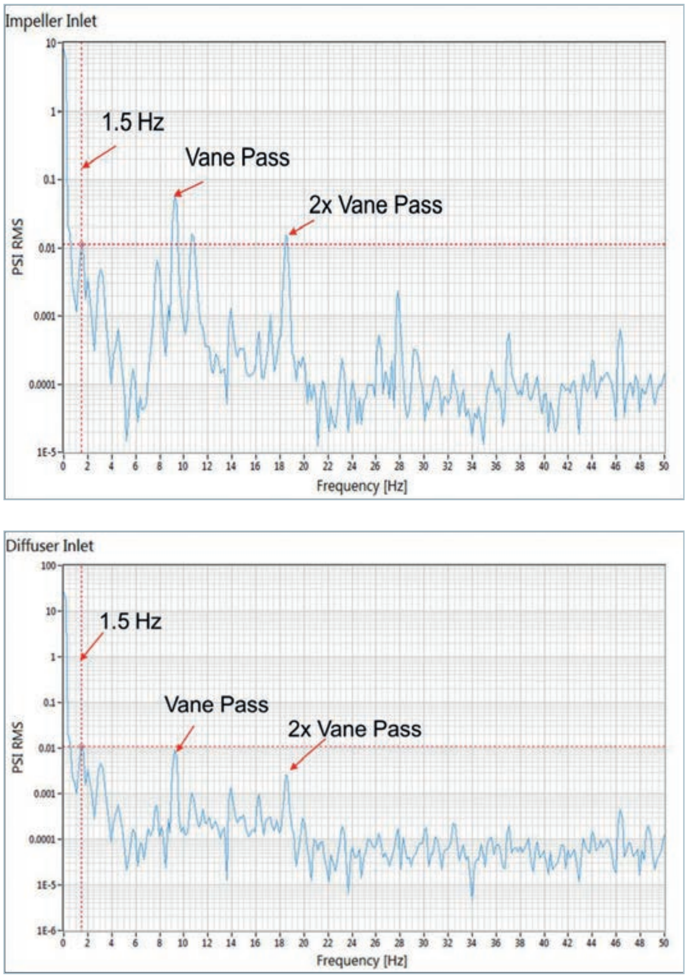


Figure 7: Relevant impeller frequencies from Simcenter STAR-CCM+ at impeller and diffuser inlets.

Simulation of a multistage pump

MSI performed simulation work on a multistage pump for a customer interested in analyzing the performance of the pump design with an unusual outlet. The centrifugal pump with multiple stages was designed to provide large amounts of total developed head (TDH). In addition to complex transient flows, rotating machinery and unsteady forced response, the pump also involved complex secondary flows, making it paramount the simulation tool captured all these relevant physics in the analysis. The pump simulation involved the flow from the inlet going through two stages with two impellers in series and discharging through a volute. A thrust balance device redirected a part of the exit flow back to the inlet. The pump had a complex primary and secondary flow path and was designed to enhance cavitation and hydraulic performance.

The flow domain of the multistage pump included the inlet, discharge, primary and secondary flow passage and interfaces connecting the stationary and rotating regions. Figure 8 shows the geometry and computational mesh on the pump. A total of eight primary and secondary interfaces connected the stationary and rotating regions in the simulation. Simcenter STAR-CCM+ was used to solve the flow around the impeller and the cavity regions in the front and rear of the shroud for each element, in addition to the flow inside the tubing connecting the inlet to the outlet.

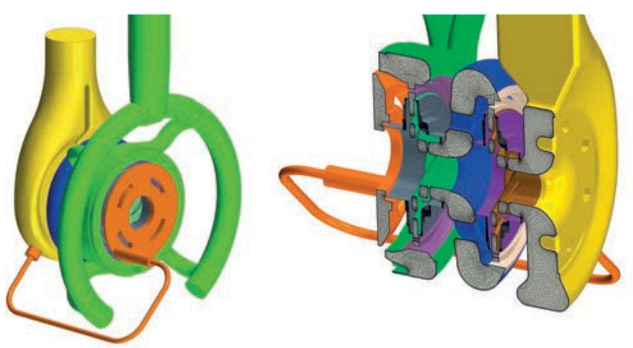


Figure 8: Geometry (left) and computational mesh (right) of pump in Simcenter STAR-CCM+.

Figure 9 shows the comparison of flow streamlines at a low- and high-flow rate through the pump. At low flow, the left side of the volute shows different flow characteristics compared to the right, where the flow is being choked off and nonsymmetrical on the left side. At high flow, the flow is vortical but more uniform on both sides. Figure 10 shows the total head from the pump at different flow rates. As observed by the customer, the change in total head levels off as the flow rate moves from high to low value. From the Simcenter STAR-CCM+ simulations, the nonuniform flow was identified as the cause of performance deterioration at low flow rates. This analysis and observation was useful for the customer in redesigning the pump for producing proper head at lower flow rates.

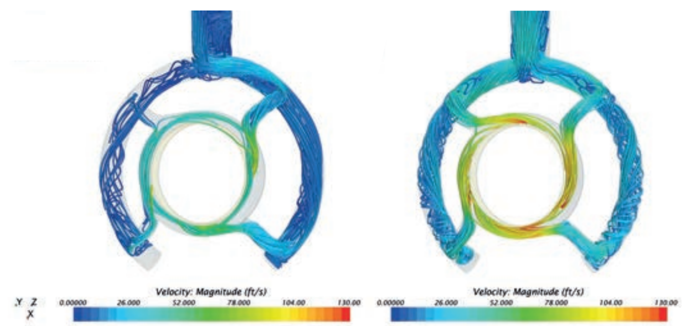


Figure 9: Flow streamlines in pump discharge colored by velocity magnitude at 359 (left) and 1,110 (right) gallons per minute.

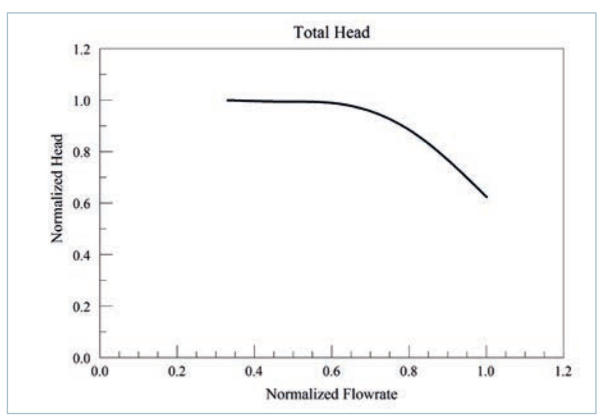


Figure 10: Comparison of flow rate (gallons per minute) versus head in Simcenter STAR-CCM+.

Conclusion

MSI has successfully incorporated Simcenter STAR-CCM+ into its process chain as a design and troubleshooting tool with desired results. The advanced, accurate flow solver in Simcenter STAR-CCM+ offers MSI a streamlined, cost-efficient engineering process to solve extremely complex problems in hydraulic turbomachinery. The case studies described here provide testimony to the benefits of numerical simulation using Simcenter STAR-CCM+ for turbomachinery design and analysis in an industrial setting. Using Simcenter STAR-CCM+ at MSI had many benefits in turbomachinery design including:

- Winning new business
- Satisfying existing customers
- Designing better performing pumps faster
- Gaining more insight compared to physical tests

Siemens PLM Software

Headquarters
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 972 987 3000

Americas
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 314 264 8499

Europe
Stephenson House
Sir William Siemens Square
Frimley, Camberley
Surrey, GU16 8QD
+44 (0) 1276 413200

Asia-Pacific
Suites 4301-4302, 43/F
AIA Kowloon Tower,
Landmark East
100 How Ming Street
Kwun Tong, Kowloon
Hong Kong
+852 2230 3308

About Siemens PLM Software

Siemens PLM Software, a business unit of the Siemens Digital Factory Division, is a leading global provider of software solutions to drive the digital transformation of industry, creating new opportunities for manufacturers to realize innovation. With headquarters in Plano, Texas, and over 140,000 customers worldwide, Siemens PLM Software works with companies of all sizes to transform the way ideas come to life, the way products are realized, and the way products and assets in operation are used and understood. For more information on Siemens PLM Software products and services, visit www.siemens.com/plm.

About the authors

Edward Bennett and Artem Ivashchenko of MSI provided the content for this white paper.

www.siemens.com/plm

66944-A3 12/17 B