Computer Simulation of Mechanical Seal Leads to Design Change that Improves Coolant Circulation

By Ray Clark and Henri Azibert
A. W. Chesterton Co., Stoneham, Massachusetts

Lanre Oshinowo, Fluent Inc. Lebanon, NH

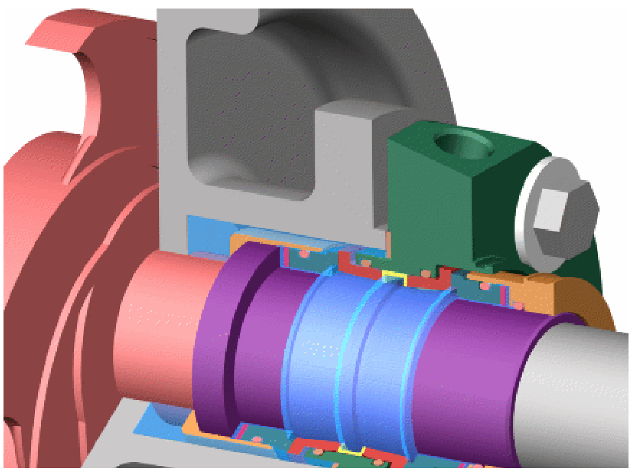

Fig 1. Centrifugal pump/mechanical seal configuration with barrier fluid flow domain.

Engineers at A.W.Chesterton Co. used computer simulation to improve the design of a mechanical seal and extend the performance limits established through laboratory testing. Using Computational Fluid Dynamics (CFD)software to simulate the flow of fluid within the seal, it was determined that the barrier fluid,which also functions as a coolant, was not circulating to areas of the seal where heat is generated.

A number of design changes were evaluated using CFD as a guide toward improving the axial circulation. Results showed that by tapering the bounding surfaces of the stationary seal rings and the shaft sleeve, axial movement of barrier fluid could be improved and heat removal increased by almost 50% compared to the earlier design. This change is now being implemented in some of Chesterton's product line, providing customers with a cooler-running, longer-lasting mechanical seal for centrifugal pumps and mixers.

Chesterton is comprised of three business units: Fluids Sealing Systems, whichincludes mechanical seals, mechanical packing and gasketing, and process pumps; Hydraulic/Pneumatic Sealing Devices; and Technical Products, which includes maintenance chemicals, and ARC component materials. Many of the commonly accepted technologies in fluid sealing systems have originated in the company's R&D labs, including such industry-changing developments as the original cartridge seals and the first commercial split seal.

Mechanical seals

Mechanical seals are used extensively in a wide variety of industries to prevent leakage from fluid handling equipment such as centrifugal pumps and mixers (Figure 1). Typically, a mechanical seal consists of two sealing rings, one made of a soft material such as carbon graphite and one made of a harder material such as silicon carbide. One ring rotates with the pump shaft while the process fluid is pumped. The other ring remains stationary. The interface between the two rings establishes the seal, preventing the process fluid from leaking. The

Chesterton mechanical seal, whose performance limits were increased for application in more severe services, is a dual seal with two sets of seal rings (Figure 2). It includes a barrier fluid, operating at higher pressure than the process fluid, which circulates within the seal. The barrier fluid helps prevent process fluid leakage between the seal rings and also acts as a coolant. It is pumped from a separate tank through an inlet port into the seal. The fluid then circulates through the seal, picking up heat. It is removed from the seal via an outlet port and flows back into the tank, forming a closed system. The tank is cooled by either natural or forced convection.

Heat is often detrimental to the lifespan of a mechanical seal, yet quite a lot of heat can be generated, since the rotating and stationary sealrings are in contact with each other. If the seal is not cooled adequately, the rubbing surfaces can thermally distort, causing the unit loads to concentrate in a few areas rather than distribute uniformly over the entire interface. In this way, localized interfacial contact can cause excess wear and decrease the life of the seal. When overheating was noticed, under extreme operating conditions, during lab testing of one of Chesterton's heavy-duty cartridge dual seals, engineers decided to study the situation. The goal was to find design changes that would improve the cooling performance of the seal, thereby broadening its performance envelope.

Problems with existing design

The first step was to understand exactly what was happening with the coolant within the existing seal. This sort of insight was impossible to obtain from routine laboratory testing, since such tests provide only the temperatures of the fluid at the inlet and outlet ports. Fluid temperatures inside the seal, as well as internal flow patterns of the fluid, cannot be easily found. To get this kind of information, engineers turned to CFD simulation, which provides fluid velocity, pressure and temperature values throughout the solution domain for time-dependent problems with complex geometries and boundary conditions. As part of the analysis, a designer may change the geometry of the system or the boundary conditions such as inlet velocity, flow rate, etc. and view the effect on fluid flow characteristics. CFD is an efficient and effective tool for generating detailed parametric studies, significantly reducing the amount of experimentation necessary to develop a device. The software used for the CFD analysis was FLUENT from Fluent Inc., Lebanon, New Hampshire. Chesterton chose this software because it handles the widest range of applications of any CFD code, including irregular geometries, specialized radiation, multiple species, turbulence,and other complicated physical phenomena.

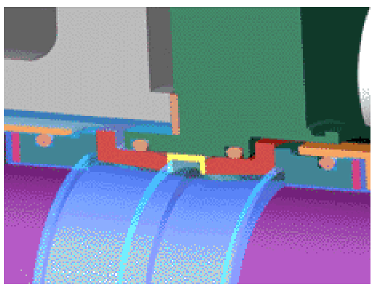

Fig 2. Cross-section of dual mechanical seal.

Engineers simulated a seal for a 48-mm diameter centrifugal pump shaft. The operating conditions were assumed to be

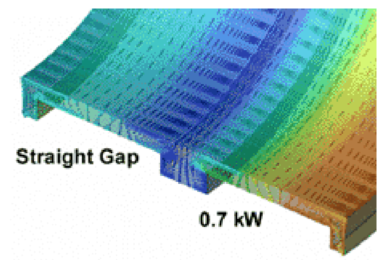

Fig 3. Axial circulation of barrier fluid for typical (untapered) design.

687 kPa and 66 degrees C for the pump process fluid and 1,031 kPa and 38 degrees C (inlet temperature) for thebarrier fluid. The variables studied included the radial clearance between stationary and rotating boundaries, taper angle of flow control surfaces, shaft rotational speed, barrier fluid through flow, and key thermophysical properties of the fluids. The FLUENT simulations involved 3D models with approximately 160,000 cells. Convergence of the analyses typically required between two and three thousand iterations. Postprocessing of simulation results was performed using specialized visualization software. All of the CFD and data visualization analyses were run on Silicon Graphics workstations.

Results of the simulations clearly indicated how the seal cooling could be substantially improved. Barrier fluid enters the seal at the inlet port via a flow channel, which is axially centered between the heat

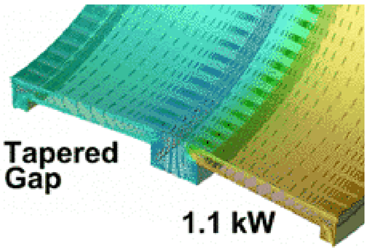

Fig 4. Axial circulation of barrier fluid for tapered surface design.

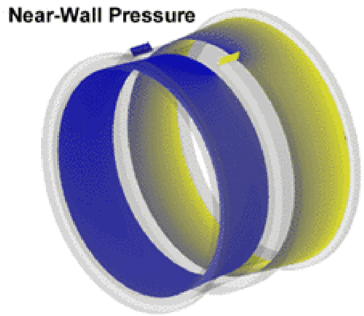

Fig 5. Pressure surfaces near seal ring (background) and rotating sleeve.

sources. Cooling efficiency, therefore, is dependent on how well the fluid circulates axially to the heat-producing interfacial positions. A graphic presentation of the FLUENT results consisting of color-coded vectors indicating fluid temperature, as well as velocity magnitude and direction of flow (Figure 3), revealed limited circulation to these critical regions of the seal.

Improving circulation

The next step was to find ways to improve axial circulation of the barrier fluid.For each modification considered, FLUENT simulations were used to evaluate the effectiveness. The change having the largest effect on axial circulation was tapering the bounding surfaces of the stationary seal rings and the shaft sleeve. Axially tapered surfaces propel cool fluid from the flow channel toward the heat-producing regions of the seal. Compared with the traditional non-tapered design, the tapered surfaces were shown to be far more effective at promoting axial flow, as evidenced by the color-coded flow vectors (Figure 4). As expected, increasing axial flow also resulted in better heat removal. The modified design showed an increase in heat removal of about 50% compared to the traditional configuration (1.1 kW heat removal for the new design versus 0.7kW heat removal for the original design).

Engineers next used FLUENT to determine the physical mechanism responsible for the improved performance of the tapered surface design. Simulation results showed that the fluid near the shaft sleeve experiences a strong centrifugal force directed radially outward from the center of rotation (Figure 5).

In the case of the tapered-surface design, this radial load has an axially directed component that drives fluid away from the flow channel toward the ends of the domain where fluid sealing/heat generation occurs. In addition, a graphical presentation of the turbulent kinetic energy of the flow (Figure 6) showed regions of relatively high turbulence near the sealing interface and near the inner, rotating wall, closer to the flow channel. This situation is advantageous since the higher turbulence and increased mixing help promote heat transfer where it was needed most.

Understanding through visualization

To better understand the nature of the axial exchange of fluid and associated thermal energy, trajectories of fluid particles released near the flow inlet were computed and displayed as streamribbons colored by fluid temperature. The flow trajectories were then animated, with twisting ribbons (Figure 7) indicating the local level of turbulence (vorticity).

The animation provided a detailed three-dimensional perspective of the helical flow patterns characteristic of the tapered-surface design. One of the techniques used allows the observer to travel the route of a fluid particle. The presentation displays the local speed and temperature of the particle, as well as the elapsed time from its release. At about 55 ms into the animation (Figure 8), the temperature-mapped trajectories, induced by low pressure, can be seen bypassing the outlet en route to the interface regions of the seal. About one-twentieth of a second later

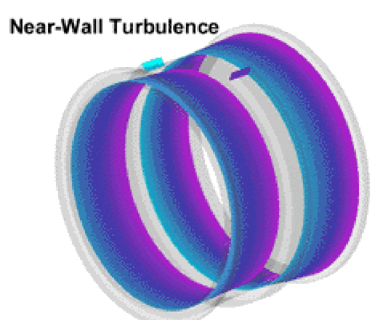

Fig 6. Surfaces of turbulent kinetic energy near seal ring (background) and sleeve.

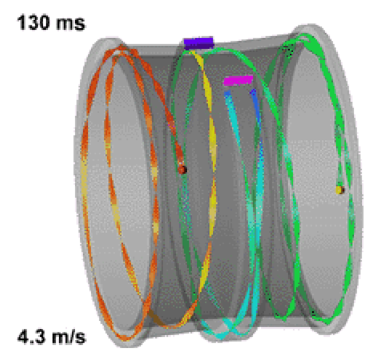

Fig 7. Streamribbon trajectories of fluid particles released near barrier fluid inlet.

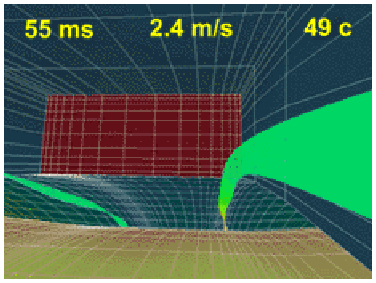

Fig 8. Scene showing streamribbon trajectories bypassing barrier fluid outlet.

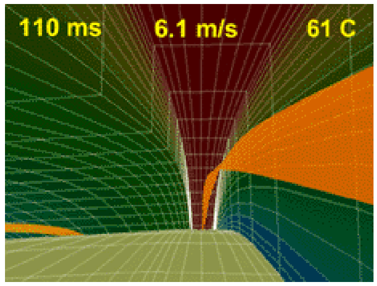

Fig 9. View of streamribbon trajectory near seal interface region.

(Figure 9), one of the paths of the accelerated particles traces alongside the rotating (radial) end wall of the domain. At this point, the particle has absorbed heat resulting in a temperature rise of 12 degrees C. After absorbing heat from the sealing interface, it reaches a maximum temperature of 62 degrees C attained along its path. During the return trip to the flow channel, the speed of the particle drops by about 25% as it traverses its way near the inner boundary of the stationary seal ring.

To validate the accuracy of the CFD results, physical experiments were conducted in Chesterton's seal test laboratory. The measured data were then compared with the FLUENT analyses. A representative case showed the predicted fluid temperature in the flow channel region for the tapered surface design to rise from the inlet to the outlet by approximately about 11 degrees C (Figure 10). The corresponding lab data (Figure 11) indicated a temperature rise of 10 degrees C, within 1 degree C or 10% of the CFD simulation.

Practical Implications

This work led to design changes in some of Chesterton's mechanical seals. It was not possible to implement the tapered surface design on all mechanical seals because a certain amount of radial space is needed to provide the taper. Some pumps don't have enough space to permit this. The seals that do incorporate the new design have performed well in the field, operating at cooler temperatureswhich should result in longer seal life. CFD simulation played a key role in bringing this improved seal to market. Without computer simulation, engineers would not have known of the problem with axial circulation in the original design, nor would they have had proof that improving it would promote removal of friction-generated heat. CFD led them to an effective design change that might not have otherwise been considered.

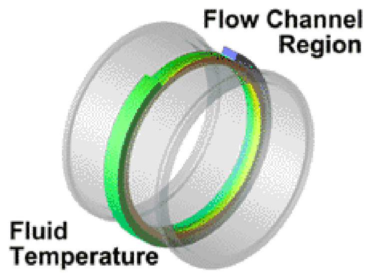

Fig 10. Barrier fluid temperatures computed for flow channel region.

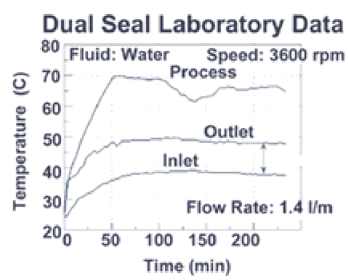

Fig 11. Laboratory data validating CFD results