OPTIMIZING SUCKER ROD COMPONENTS IN ROD-LIFT SYSTEMS: LEVERAGING COMPUTATIONAL FLUID DYNAMICS (CFD) TO ENHANCE DESIGN AND RELIABILITY

Jesus Abarca MsC, Esteban Oliva, and Matías Pereyra

Tenaris

ABSTRACT

Rod-lifted wells in U.S. unconventional fields have been pushed beyond their limits since the onset of the unconventional reservoir (UR) revolution. Sixteen years later, the demand for higher production rates with rod-lift systems remains strong. As the industry progresses toward the Aspirational Goal of 1,000 barrels per day (bfpd) at depths of 10,000 feet (1K @ 10K), new challenges continue to emerge.

While previously identified issues, such as wellbore deviation, high sideloads, and compressive loads, have been mitigated through innovative rod guiding techniques (Oliva & Anderson, SWPSC 2024, Sinker Section Design to Reduce Buckling-Related Failures), operators in the 400 to 600 bfpd range now face additional challenges. Specifically, turbulent flow conditions have led to corrosion-erosion mechanisms around rod guides and connections.

This study explores the use of Computational Fluid Dynamics (CFD) as a tool to enhance the design and reliability of sucker rod components in rod-lift systems. By applying CFD techniques to model fluid dynamics, we optimize key properties of rod guides and connections, such as geometry, dimensions, and Erodible Wear Volume (EWV). This approach allows for precise optimization of component placement and design, ultimately improving runtime and reducing wear-related failures in challenging operational conditions.

BACKGROUND

A vast number of engineering applications can nowadays be simulated through Computational Fluid Dynamics. In the oil and gas industry, such applications range from simple internal flow analyses in pipes to complex evaluations of multiphase fluid behavior, including particle transport through fittings or heat exchangers. In artificial lift, CFD simulations help us understand and compare the fluid dynamic behavior of geometries such as a single guide or multiple guides placed in sequence, as well as connections or a combination of these.

In the same way, CFD simulations assist in defining geometries with features and characteristics that minimize fluid disturbance and drag loads while extending their operational lifespan. Iteratively combining CFD simulations with FEA stress analysis enables the design of more efficient fittings with improved mechanical strength and optimal fluid-dynamic performance.

In Rod Lift, CFD models are also used to analyze erosion patterns in connections, couplings, and forged ends, as every shape and section change in a component alters fluid dynamics, as well as fluid serve as a medium for transporting particles such as sand and steel oxides from rod corrosion, which can erode any section of the rod string or the tubing. Features on string components that generate fluid eddies can be identified and modified or improved when they correlate directly with erosion-corrosion failures.

THE PROBLEM

In rod lift systems, fluid dynamics plays a crucial role in corrosion-erosion by intensifying turbulence and shear stresses within the tubing. High-velocity multiphase flow disrupts protective corrosion films (passivation), chemical inhibitor films, exposing metal surfaces to CO_2, H_2S, and chlorides. Additionally, sand, scale and corrosion byproduct particles carried by the flow accelerate mechanical wear. In wells with high gas-to-liquid ratios or water cuts, rapid fluid motion increases both corrosion rates and erosion damage, particularly in areas with restrictions, bends, or pump intakes.

Localized turbulence and high-speed fluid impacts, especially at rod-tubing contact points, further exacerbate material loss. Liquid slugs and rod reciprocation generate impingement forces, leading to severe thinning of tubing walls and sucker rods. The alternating exposure to corrosive fluids and erosive forces accelerates degradation, compromising the structural integrity of well components.

To mitigate these effects, operators optimize production rates, control sand influx, and use corrosion-resistant materials. Chemical treatments, coatings, and plastic-lined tubing help reduce metal degradation. Proper rod string alignment with centralizers also minimizes turbulence and wear. Managing fluid dynamics effectively extends equipment life and reduces costly failures.

Tenaris has performed several Root Cause Analysis in equipment coming out from wells in the US, identifying sucker rod failures showing evidences of corrosion erosion mechanisms at the discharge of rod guides or around the rod connection.

In the other hand, operators' Life of Well production strategy is defined by a myriad of parameters. Among the different paths that lead to Rod Lift (RL), the most common is an initial stage on Electrical Submersible Pumps (ESP) that can last between 1 and 4 years in the Permian Basin UR wells, depending on productivity index. One of the major differentiators between ESPs and RL is the repair cost, typically being the latter one more cost effective, especially within comparable production rates. As the well production declines, operators face the decision of at what point is it more convenient to migrate from ESP to RL, of course, making sure to satisfy the basic premises of maximizing production and minimizing expenses. Rod Lift suppliers are making strong efforts to help pushing the upper limits of the system by upsizing some of the main system components: Pumping units and subsurface pumps. Now, the piece that connects these two (sucker rod string) must satisfy a list of complex features:

- Produce above 350bfpd @ approx. 9000ft pump landing depth
- High effective stresses, coming from multiaxial loads
- Heavy loads to meet very deep pump locations and high flow rates
- Aggressive operating environments, with corrosive fluids
- Certain corrosion fatigue failure mechanisms… sooner or later
- Meet operator expectations of reliability
- Potential high wear rates for tubing and rods due to tribocorrosion and three-part abrasion mechanisms.
- Advanced corrosion-erosion mechanism due to high turbulent flow in multiphase conditions.

In consequence, Tenaris Rods launched a project named "1K @ 10K". What this means is that Tenaris took the mission of aiming long beyond the current limitations. Thus, this reads 1,000bfpd at 10,000ft deep. The strategy sought is that by aiming big, far, the closer goals are going to become more certain and within reach.

COMPUTATIONAL FLUID DYNAMICS OVERVIEW

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and algorithms to analyze and solve problems involving fluid flow, heat transfer, and other related physical phenomena. CFD simulations are widely applied in various industries, including aerospace, automotive, energy, biomedical engineering, and environmental science. By leveraging computational power, CFD provides engineers and researchers with valuable insights into fluid behavior, eliminating the need for extensive physical testing and experimentation.

How CFD Works

CFD works by solving the fundamental equations governing fluid motion, specifically the Navier-Stokes equations, which describe the conservation of mass, momentum, and energy. These equations, however, are highly complex and nonlinear, making analytical solutions infeasible for most real-world applications. Instead, CFD employs numerical techniques to approximate solutions by discretizing the fluid domain into smaller elements or control volumes.

Key Steps in CFD Analysis

1. Preprocessing

Geometry Definition: The physical domain of interest is modeled in a CAD software or directly within the CFD software.

Meshing: The fluid domain is divided into a finite number of small control volumes (elements or cells). The quality and density of the mesh significantly impact the accuracy and computational cost of the simulation. Adapting mesh can improve results.

Solver and Type of analysis, Pressure or Density based, transient or steady. This definition is depending of phenomena to be analyzed.**.**

Model and operating conditions, turbulence model selection, choose a model based on the flow characteristics (e.g., k-ε, k-ω, or Large Eddy Simulation), Material and media are defined

Boundary Conditions: Users define input conditions such as inlet velocity, pressure, temperature, wall conditions, and outlet constraints to simulate real-world environments.

2. Solving

Iterative solvers, such as the SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm or Pressure-Implicit with Splitting of Operators (PISO) algorithm, are used to solve for fluid flow parameters.

Computational solvers handle nonlinear equations iteratively until a converged solution is achieved within an acceptable error margin.

3. Postprocessing

The results, including velocity fields, pressure distribution, temperature gradients, and flow patterns, are visualized using software tools.

Engineers analyze flow characteristics, detect potential issues such as turbulence or recirculation zones, and optimize designs based on simulation findings.

Comparisons with experimental data or theoretical predictions help validate the CFD model's accuracy.

An extended value is to analyze drag forces

Understanding CFD Reports: Lines and Colors

CFD reports typically use various colors and lines to represent different flow characteristics and physical quantities. Here is what they generally indicate:

- **Streamlines**: Curved or straight lines that indicate the direction and path of fluid flow. They help visualize turbulence and vortex formations.
- **Velocity Contours**: Colors that represent different velocity magnitudes within the fluid domain. Higher velocity regions are usually indicated by warm colors (red, orange), while lower velocity regions appear in cooler colors (blue, green).
- **Pressure Contours**: A gradient of colors showing pressure variations. High-pressure zones are often marked in red, while low-pressure areas are in blue.
- **Temperature Distribution**: A color map displaying temperature variations across the domain, useful in heat transfer analysis.
- **Turbulent Kinetic Energy (TKE)**: A measure of turbulence intensity, indicating the distribution of energy within turbulent eddies. Higher TKE values represent areas of increased turbulence and mixing.
- **Vectors**: Arrows representing the magnitude and direction of flow at specific points, helping in understanding fluid motion and potential recirculation zones.
- **Iso-surfaces**: 3D surfaces of constant value (e.g., constant temperature or pressure) used to visualize spatial distributions of fluid properties.

Applications of CFD

- **Aerospace**: Optimization of aircraft aerodynamics, reduction of drag, and thermal management.
- **Automotive**: Improving vehicle aerodynamics, engine cooling, and fuel combustion efficiency.

- **Energy**: Enhancing wind turbine efficiency, analyzing thermal performance in power plants, and simulating oil and gas pipelines.
- **Biomedical Engineering**: Studying blood flow in arteries, designing medical devices, and optimizing drug delivery systems.
- **Environmental Science**: Predicting pollutant dispersion, weather modeling, and simulating ocean currents.
- **Oil and Gas Wells**: CFD is used to optimize drilling operations, study multiphase flow behavior in wellbores, analyze pressure drops in pipelines, improve hydrocarbon recovery techniques, and assess safety measures related to gas dispersion and blowouts.

CFD IN ROD LIFT SYSTEMS - APPLICATION

Computational Fluid Dynamics simulations in Rod Lift applications can be categorized based on the type of system, beam pumping (BP) or progressive cavity pumping (PCP), and the specific phenomena under study (Figure 1). For erosion-corrosion effects, particle size and fluid density are key factors. However, when the objective is to observe turbulence intensity, fluid velocity and detailed geometric shapes are key to obtaining best and accurate results.

Tenaris has already used this tool in the past for the development of the sucker rod guide Tenflow™ and published in 2019 SWPSC (Padron & Abarca. DEVELOPMENT OF A ROD GUIDE MODEL, WHICH GENERATES A MINIMUM LEVEL OF TURBULENCE, PERFORMING CFD ANALYSIS AND HYDRODYNAMIC COMPARISONS BETWEEN DIFFERENT GUIDE DESIGNS. Paper published in SWPSC 2019)

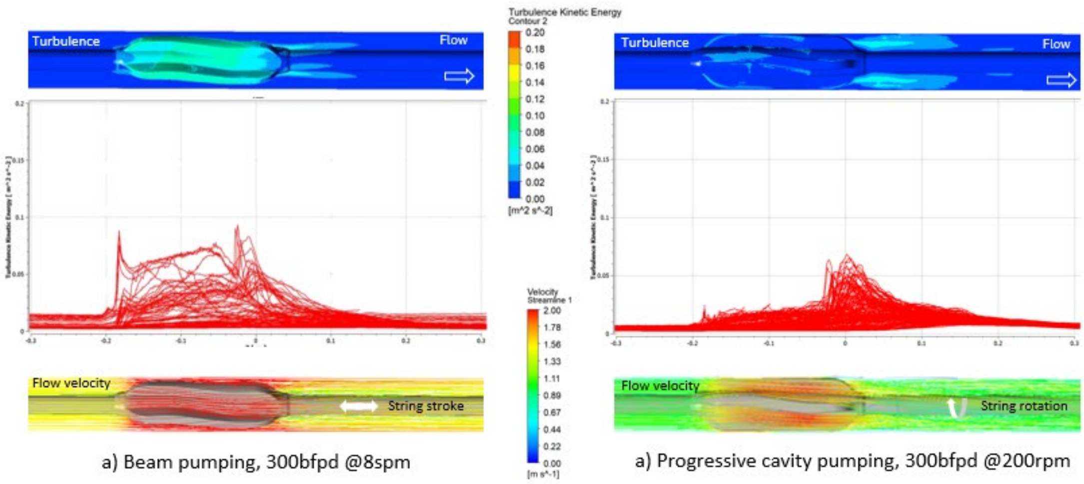

Figure 1. Turbulence comparison in BP and PCP applications with same guide

In beam pumping systems, when searching for critical conditions, the system's maximum speeds must be considered (Figure 2).

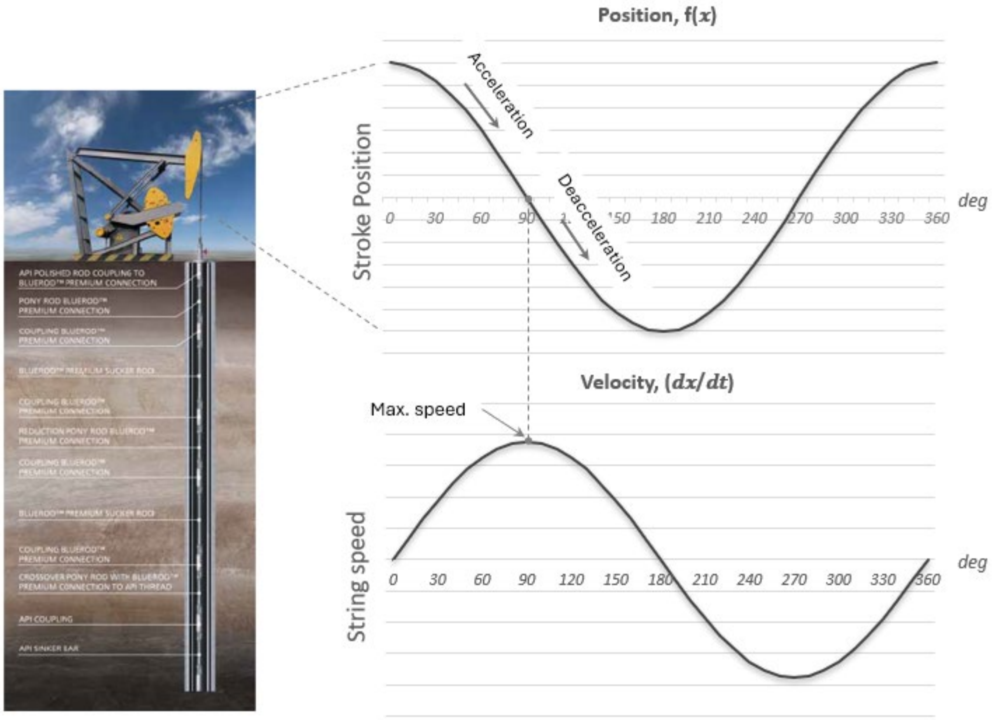

Figure 2. Idealized stroke and velocity in beam pumping systems

For systems with fiberglass rods, the elasticity of the fiber section introduces a spring effect that must be considered (Figure 3). Steel sucker rod sections exhibit slight velocity differences compared to the top of the well.

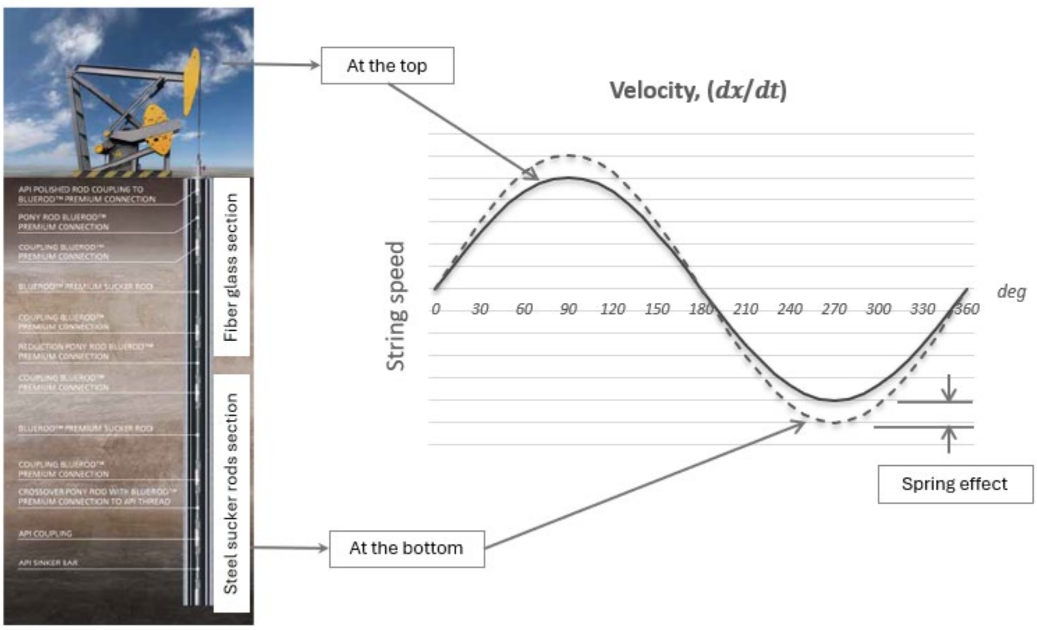

Figure 3. Idealized string velocity with fiberglass rods.

CFD also helps analyze fluid behavior and turbulence generation due to accessories in a string as it descends through the fluid (Figure 4). The effect observed during string deceleration and stroke reversal also creates turbulence and eddies, particularly in the guide section facing the surface. This suggests that not only critical descent conditions impact the removal of corrosion inhibitor films (in rod OD and tubing ID), but also the change in direction of the string and the inertia of the fluid. The transition of the guide end towards the rod body is key to managing turbulence at the stroke reversal point.

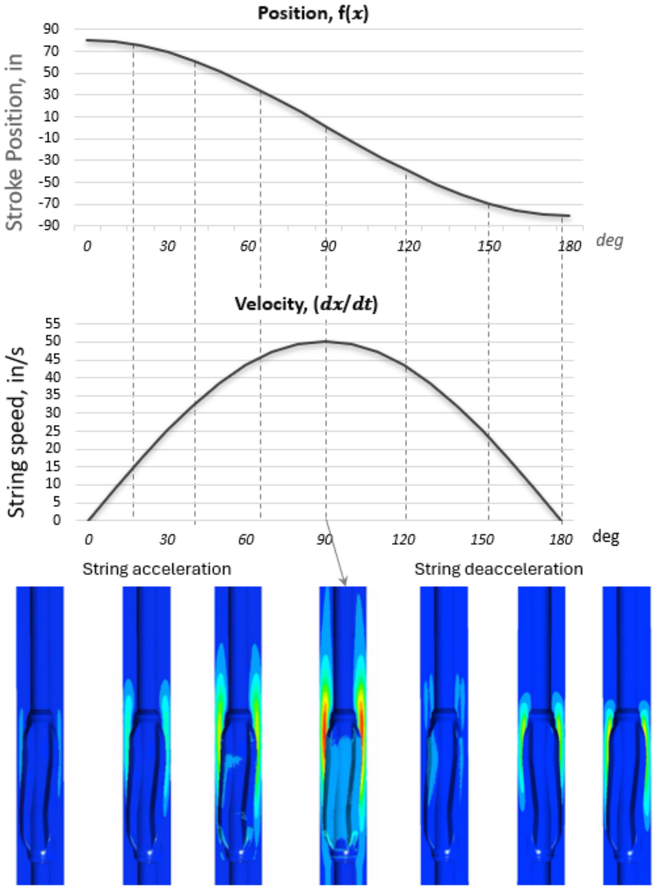

Figure 4. Turbulence levels generated by a guide during string downstroke, simulated using a transient model.

Regarding connection elements, section changes and methods to protect them from metal-to-metal wear can also be analyzed through CFD studies. Complete string sections can be analyzed and optimized to ensure guide distribution minimizes turbulence and provides an optimal connection centering avoiding the risk of contact with the tubing (Figure 5).

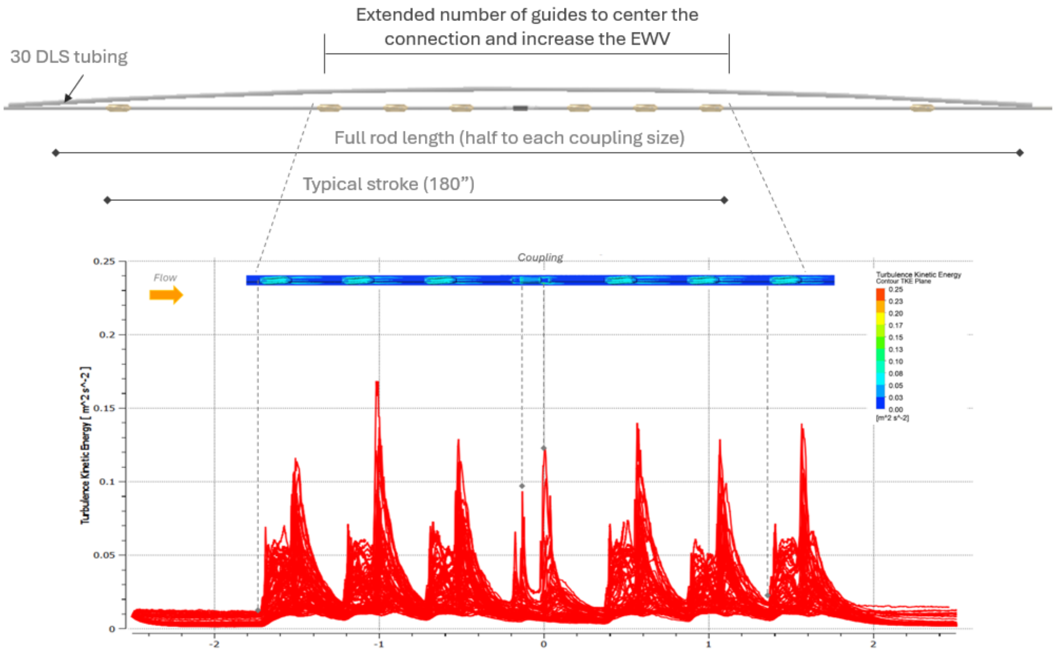

Figure 5. String guided with targeting maximum erodible wear volume concentrated near the connections.

Another important aspect analyzed and troubleshooted through CFD is the many section dimensional and geometrical changes in the connections and forged ends. The connection analyzed in Figure 6 shows that the section changes are key in mechanisms of erosion-corrosion in the square wrench, since the changes generate eddies. This compounded with the presence of viscous fluids, muds, emulsions or with high particle content accelerates the erosion-corrosion phenomenon on the flat surfaces of the square wrench. This mechanism extends to couplings, especially in the steps created by OD differences between coupling and pin end shoulder (i.e. full size couplings) (Figure 7).

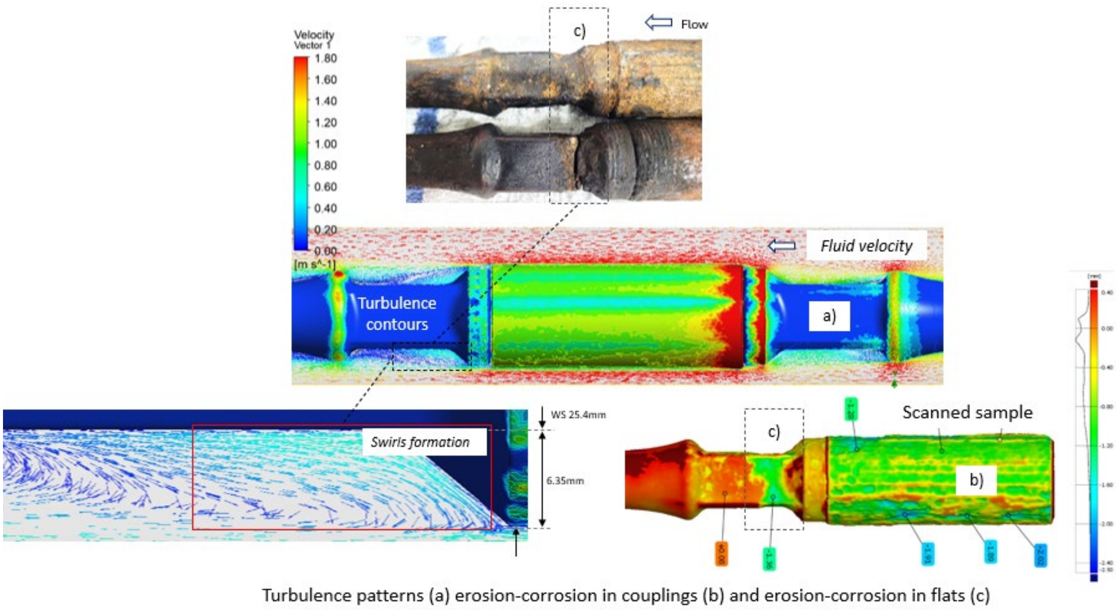

Figure 6. Erosion-corrosion patterns on full size coupling and square wrench

CFD applications extend beyond analyzing turbulence generation and fluid behavior through fittings. Connection designs can also be improved, particularly by refining section transitions in forged ends as smoother transitions reduce fluid alteration. Coupling size, whether FS or SH, with or without coating, plays a crucial role in erosion-corrosion failure risks. The histogram in Figure 7 illustrates how high turbulence exposure in full size couplings is reduced when using slim hole couplings.

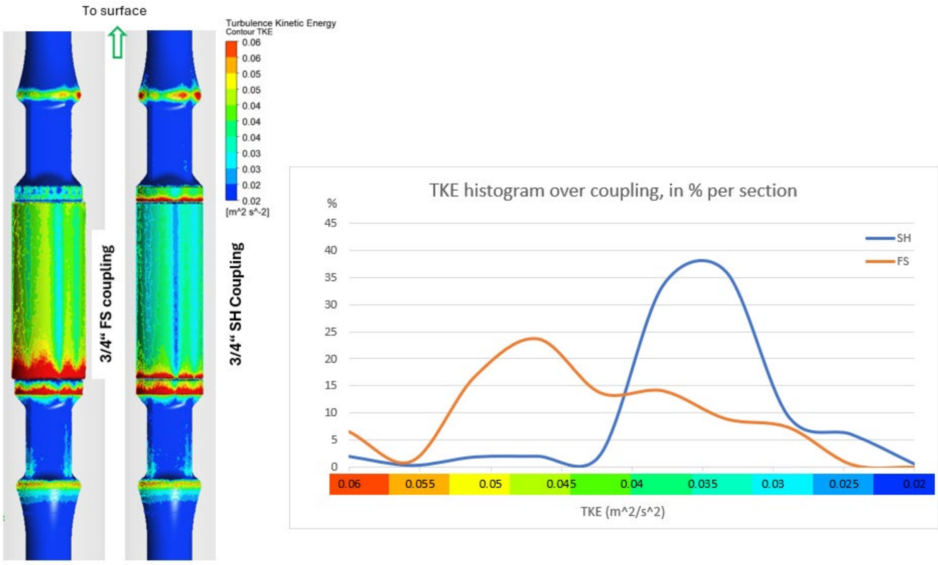

Figure 7. Turbulence comparison between FS and SH 3/4" couplings.

ROD GUIDE EVOLUTION

As indicated initially, Tenaris application of RCA process has helped to identify cases where engineering improvements over products can help mitigate failure mechanisms generated by changing operational conditions. The Tenflow® guide was designed aiming to certain operational requirements and CFD Boundary Conditions corresponding up to 320bfpd flowrate (Padron & Abarca. DEVELOPMENT OF A ROD GUIDE MODEL, WHICH GENERATES A MINIMUM LEVEL OF TURBULENCE, PERFORMING CFD ANALYSIS AND HYDRODYNAMIC COMPARISONS BETWEEN DIFFERENT GUIDE DESIGNS.Paper published in SWPSC 2019). Since Rod Lift has expanded this application range to flowrates beyond 350bfpd and exceeding 500bfpd, its design optimization was needed.

With the application of CFD analysis improved geometries can be defined to maximize performance Figure 9 depicts the comparison where vane thickness, shape and surfaces were modified to reduce flow restriction while the support footprint against the tubing is conserved. This change in geometry reduced considerably the turbulence generated by the guide, eddies length and the turbulence projected over the rod downstream the guide aiming to reduce the inhibitor film remotion (Figure 10)

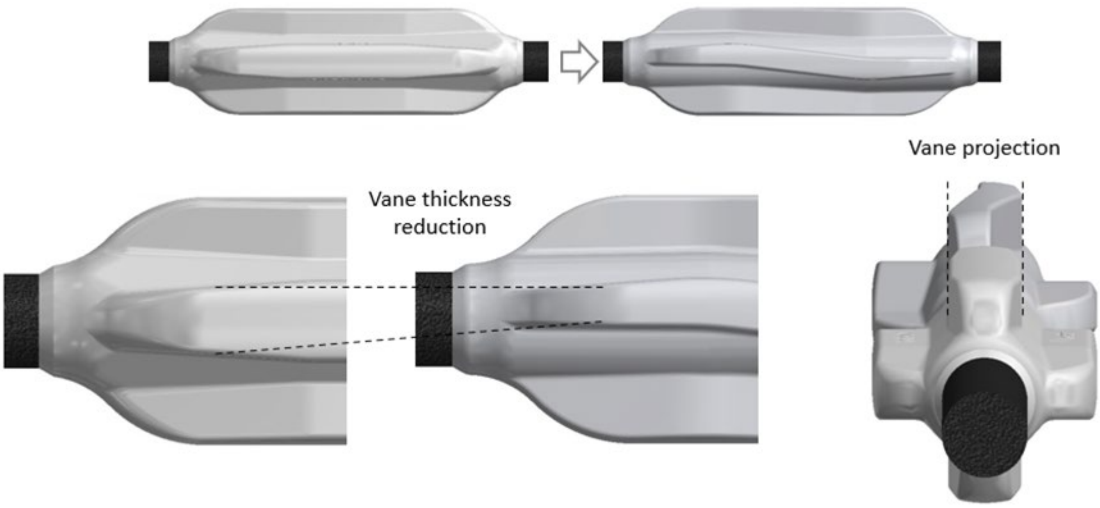

Figure 9. Geometry optimization with the use of CFD simulations

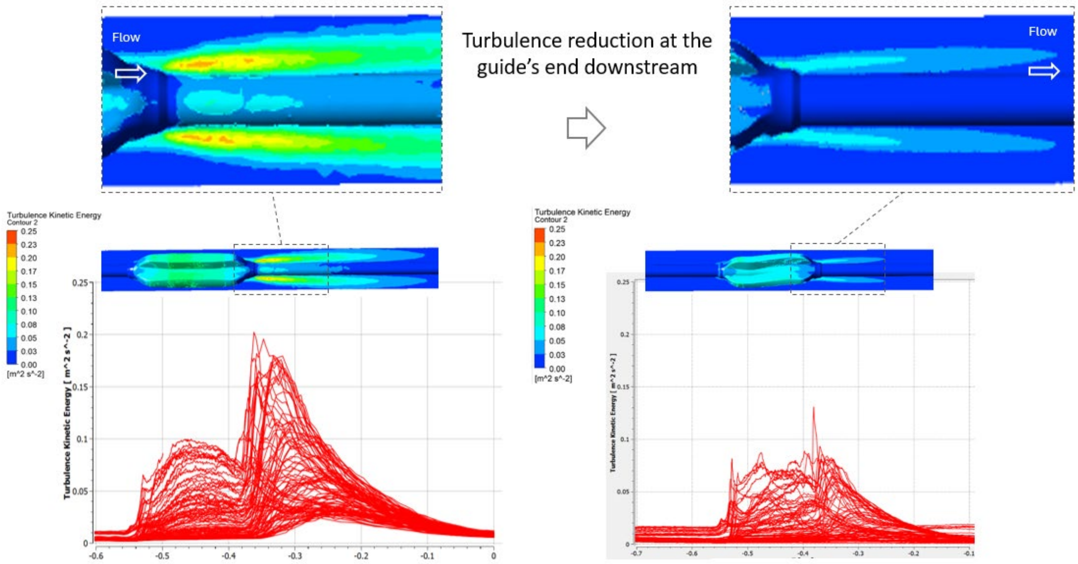

Figure 10. Turbulence comparison between guides, TenFlow™ and TenFlow™ XF

NEXT STEPS:

Multiphase CFD including gas phase

CONCLUSION

CFD has revolutionized the way fluid-related problems are addressed, offering cost-effective and highly detailed simulations that improve engineering design and performance analysis in relatively short times.

Specifically for Tenaris Rods, CFD helped during the development of the first Permian targeted rod guide (TenFlow™) in 2019 and helps continue evolution to meet more demands in performance and reliability now with TenFlow™ XF.

It has been key in the development of the Sinker Rod pin ends and the 9per Optimized for Compression configurations (Guillermo Ghione et al, FULL-SCALE TRIBOCORROSION AND ABRASIVE TESTING TO MITIGATE ROD AND TUBING WEAR, SWPSC 2025).

Finally, Tenaris Rods has made CFD a tool of continuous support to Root Cause Analyses, whenever deemed necessary.