CFD simulations of flow jetting impact and high erosion region in a production choke and its downstream spool

Agastian Perinpasivam [a,*], Nikhil Bagalkot [b], Arvind Keprate3 [b]

[a] *Westcon Yard AS Haugesund Norway,*

[b] *Department of Mechanical Engineering, Oslo Metropolitan University, Norway*
nikhilba@oslomet.no

Abstract

Erosion wear is a rather well-known problem in the petroleum and transport industry. Over the years there have been many different models suggested to estimate the erosion. Each model uses unique equations and is suited for different types of geometries, which gives different flow profiles and erosion patterns. It is critical to know where the erosion wear occurs and at what magnitude the system is located at to predict and economically create a choke valve design. The erosion study additionally, helps in accessing location and thickness of cladding required to prolong the life of components. In this study, computational fluid dynamics (CFD) is used to investigate erosion on a production choke valve and its downstream spool. There are three main steps to predicting erosion wear using CFD analysis: flow model, particle tracking and calculating the erosion wear from particle interaction. The results indicate that pressure drop affects the velocity jet shape and impact region. In the 50% opening case, the pressure drop creates a wide jet stream region that, in turn, will cause an increased wall impact region. When the opening decreases, to 35% and 25%, the jet stream gets more focused. The velocity jet impacts a smaller area of the pipe wall, which in turn creates possibilities for increased erosion rate. The high pressure drop in the 15% opening case creates a high-focused jet stream in the middle of the downstream pipe, leading to low wall interaction.

1. Introduction

For the production of oil and gas (surface and subsea), Production Choke Valves (PCVs) are used to control the flowrate and pressure further downstream. PCVs are valves designed to take the brunt of the pressure of the line components, as this helps to increase the life expectancy of the system. The primary task of the PCV is to lower the pressure to a manageable level, so that the oil and gas can be transported safely (McLaury, Shirazi et al. 2000).By restricting the flow to a very small orifice, it reduces the well pressure downstream. PCVs have the ability to change the cross-section area (throat) either manually or by using sensors. The change in the area would lead to varying of the pressure resulting in a pressure drop and increase in velocity, which makes the choke valve most subjected to erosion.

Generally, a mixture of oil, gas and water is flowing through the system, which will contain small impurities like sand particles. In most of the cases there will be sand filters installed, which are effective in removing the large sand particles. However, smaller size sand particles (<150 micrometer), end up in the production lines, which are the main reason for erosion. As the production choke reduces the pressure, the loss in pressure energy will result in increase in the kinetic energy. Therefore, the reduction in pressure will cause the flow and the particles to move up to sonic velocities. Erosion rates at these high velocities will become substantially high, which will eventually lead to repair, replacement or leakage of the components.

The damage of choke valves and the downstream production flow lines is dangerous for workers in the area, and the replacement cost is extremely high (Haugen, Kvernvold et al. 1995, Raghavendra, Shivashankar et al. 2014). Thus, the need for good materials (Wheeler, Wood et al. 2006) and design optimization is very desirable. Increasing the longevity of a choke valve will have a significant cost reduction on top of the area being more secure.

Erosion is an occurrence that happens when mass is worn away from the material surface due to either chemical or physical interaction. In the industry, any process that transports solid particles in a fluid phase is vulnerable to erosion damage (Haugen, Kvernvold et al. 1995, Oka, Okamura et al. 2005, Desale, Gandhi et al. 2009). It often occurs in line components such as pipe bends, tubes and structures that alter the flow field. Pipelines that bend cause shifts in the flow, which makes the particles hit the wall of a surface with high pressure or velocity.

Over the years researchers have spent a significant amount of time trying to understand erosion models, what type of mechanisms and material needed when developing pipeline systems (DNV 2015). Both fluid (velocity, density, pressure to name a few) and particle properties (size, shape, density, and concentration) play a critical role in erosion phenomena and has been well researched (Wallace, Dempster et al. 2004, Mathew 2017, Bishnoi, Kumar et al. 2021). The effect of particle size and shape have been a big topic of investigation with many researchers. Oka, Okamura et al. (2005) created an erosion model that defines that erosion rate caused by sand particles on stainless steel is dependent on the shape, velocity and impact angle of the particle. In their study they find that erosion rate increases as the particle size becomes bigger. Similarly, Feng and Ball (1999) came to the conclusion that particle size increases the erosion rate. The size affects both the particle impact velocity and the kinetic energy.

In addition to fluid and particle properties, the design of the production choke plays a critical role. Changes in valve geometry can impact the erosion wear significantly. Given time, erosion can drastically change the flow field, which can affect production and later result in equipment failure. McLaury, Shirazi et al. (2000) explains this in a computational study where the effects of erosion wear and geometry changes were compared between experimental erosion results and an erosion prediction model. A waterflow mixed with sand particles was directed through a choke geometry with a sharp entrance profile, and the rounding of the leading edge began immediately after starting the test. In addition, the turbulent kinetic energy near the entrance was large which, in turn, led to turbulent fluctuations in the flow. As a result of the high turbulent fluctuations, more sand particles struck the geometry wall, causing increased erosion. Results showed that the predicted erosion rates were larger than the experimental results, but when the rounded edge was accounted for, the predictions matched the experimental results very well. This shows the importance of accounting for changes in the geometry to attain accurate predictions.

Wallace, Dempster et al. (2004) investigated the capability of computational fluid dynamics techniques to estimate the erosion rate in two different valve geometries, a simple geometry with basic geometric features and a more complex choke geometry. Measurements from a parallel experiment were used as comparison regarding erosion rates and flow coefficient predictions. The test resulted in underestimated erosion rates; however, erosion location matched the experimental data, along with the flow characteristics. It is suggested that neglecting the changes in the model geometry due to erosion could be one contributing factor to the faulty erosion rate predictions.

The main objective for the current work is to simulate erosion in a PCV. In the industry, and in previous research on this topic it is always assumed that the maximum erosion or the erosion hot spot is mainly at the first “U” bend downstream of the choke. Therefore, there is a thicker cladding on the “U” bend section than other regions. Some of the analytical models are also based on this assumption. The novelty of the current work is identifying the hot zones (high erosion zones) with respect to opening of the choke valve and mass flow rates. The findings of the study will help in accessing where the cladding is necessary. The analysis will cover the danger zones on three critical parts of the choke valve, the needle, the seat, and the downstream piping region. To achieve this, we will also need to simulate multiple high speed, compressible flow cases. We will discuss the relation between downstream jet impact regions and the pressure drop. Finally, we will compare erosion impact areas between the cases and see which cases produce the highest erosion rates.

2. Methodology

Figure 1 shows the methodology employed in the present study. The study starts with the gathering of a CAD model and simplifying it to suit the CFD simulations; this is followed by meshing and finally CFD simulations. The following section 2.1 to 2.3 will give a detailed description about each of the steps mentioned in Figure 1.

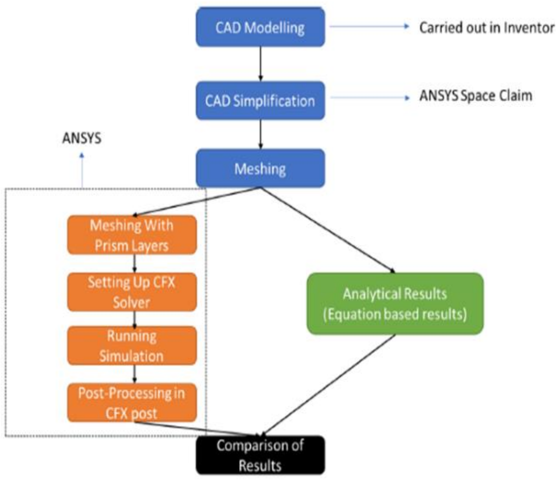

Figure 1: Flowchart of the methodology.

2.1. CAD Model Simplification

The model shown in the Figure 2A, was obtained from open-source platform called GrabCAD. A needle seat type production choke valve is used to carry out the present study. The needle and seat valve is adjustable, which means it can be used to change and control the flow and the pressure parameters. When the choke is completely closed, the needle sits tight into the seat, restricting all flow. The current study investigates a valve where the inlet and outlet dimensions are 2” and 1”, respectively. When simulating a flow field, the important part of the valve geometry is the internal volume. The internal volume was isolated, and all other unnecessary parts of the valve were discarded. The inlet and outlet regions were also extended to ensure fully developed flow through the choke valve as seen in Figure 2B.

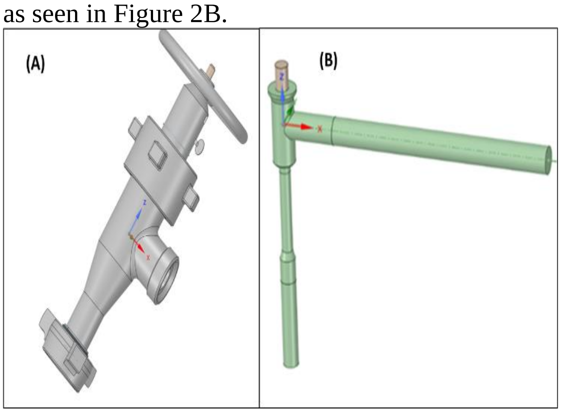

Figure 2: Original and Simplified CAD model.

2.2. Meshing

A finite volume unstructured meshing was used to mesh the CFD domain shown in the Figure 2B using the inbuilt ANSYS Meshing tool. To decide the number of the mesh elements and to get accurate results using CFD, a mesh sensitivity analysis is necessary. The idea is to run the same simulation with different mesh element sizes and compare the results. When the results do not change even though the number of elements is increasing, the desired mesh size to continue the analysis is found. CFX settings for mesh sensitivity analysis, were to set steady state, with air ideal gas at 25 degrees Celsius. A mass flow intel rating 25 kg/s, and a static pressure outlet that is 200 bar.

The analysis consisted of five different mesh sizes with 1.5, 3.1, 4.6, 6 and 8.1 million elements, respectively. Maximum velocity at a fixed point in the downstream region and maximum velocity at inlet was compared between the different meshes and results are shown in the Figure 3 below. As the graph shows (Figure 3), the values do not change noteworthy after 4.6 million elements, so choosing a finer mesh would result in increased computational requirements without increased accuracy. Further, inflation layers were created, whose first layer size is driven by the size of the particle to be analyzed. The particle size used in the present work is $100*10^{-6}$ m, hence the first layer inflation size is $105*10^{-6}$ m, with 10 layers and growth factor of 1.2. Figure 4 shows the final mesh with inflation layer distribution.

2.3. CFD Simulation and Validation

A transient CFD simulations was carried out using ANSYS CFX. A SST turbulence model was employed and the simulations were carried out for four opening of the valve (15%, 25%, 35% and 50%) and for two flow rates of gas (2.5 kg/s and 12.5 kg/s). A minimum of 30,000 particles were injected. The erosion model and its prediction is based on the DNV method (DNV 2015). First, Steady state simulation (only flow) was carried out, the results of the steady state simulations was then used as initialization for the transient state simulation (flow and sand injection), which was run for 1 second, with a max time step of 0.001s and minimum timestep of 0.00001s. During the transient simulations the sand particles were injected at the rate of 100000 per second. After one second the sand injection was stopped and simulation was run until at least 90% of the sand particles were traced back at the outlet.

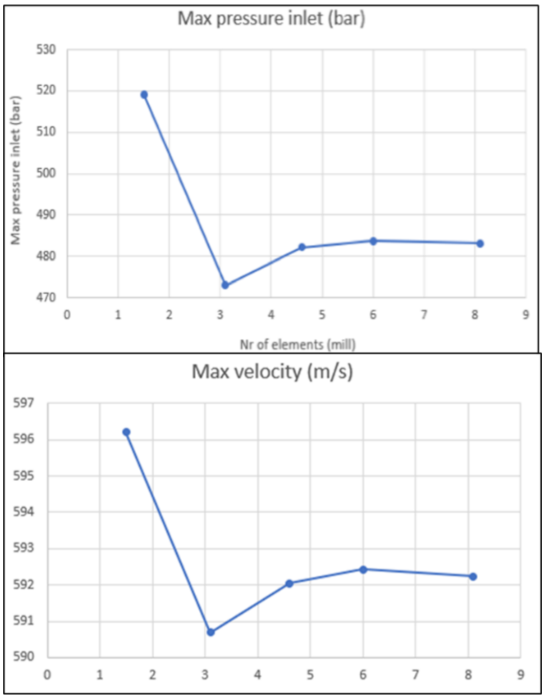

Figure 3: Mesh sensitivity analysis.

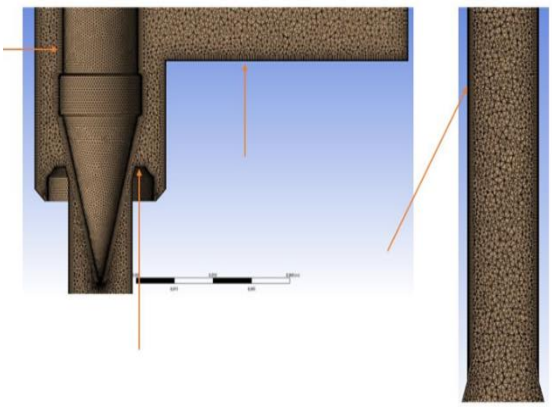

Figure 4: Mesh Distribution on the model.

3. Results

In this section results are discussed. Figure 5 shows the results for 50% opening at 2,5 kg/s mass flow rate showing the results of pressure, velocity, and particle tracks. The results show a pressure drop around the needle tip, causing the velocity to increase. At 50% opening, the velocity jet stream spreads the particles downstream, causing a wide impact region, which is different than the results are for 15% opening at 2,5 kg/s mass flow rate shown in Figure 6. At 15% opening there is greater pressure drop around the needle tip compared to 50% opening, causing the velocity to increase. The velocity jet stream carries the particles far downstream with a focused jet shape, causing almost none wall impact. Therefore, flow details the possible hot spots for the erosion to occur.

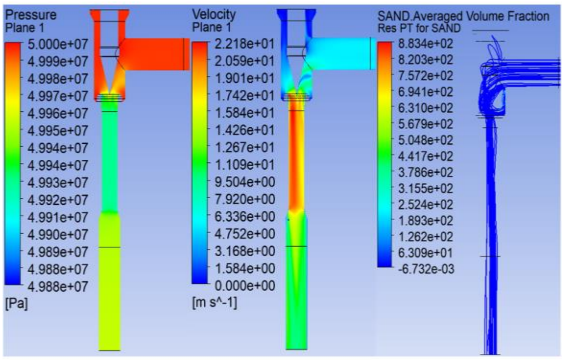

Figure 5: Pressure, velocity, and particle tracks for 50% opening and 2.5kg/s mass flow rate.

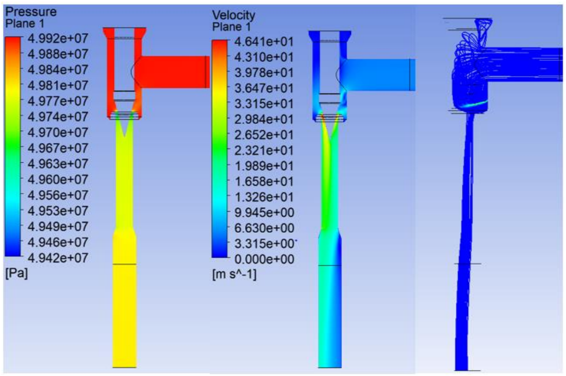

Figure 6: Pressure, velocity, and particle tracks for 15% opening and 2.5kg/s mass flow rate.

Figure 7 and 8 takes the analysis into erosion estimation. As mentioned before, the erosion estimation was carried out using DNV model, with the inputs like velocity and angle of impact calculated by the CFD analysis. Figure 7 shows the results are for the downstream region at 50% opening and 2,5 kg/s mass flow rate and shows erosion rate density at user defined values. It may be observed that most of the erosion happened immediately downstream of the choke, as predicted by the velocity results. Similar, Figure 8 shows the erosion profile for 15% opening at 2.5 kg/s. It may be clearly observed that the jet does not significantly impact the downstream walls. Again inclining with the velocity observations.

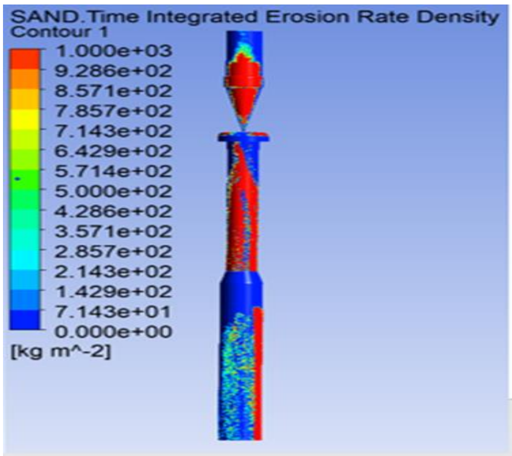

Figure 7: User defined erosion values at 50% opening and 2.5 kg/s mass flow for entire geometry.

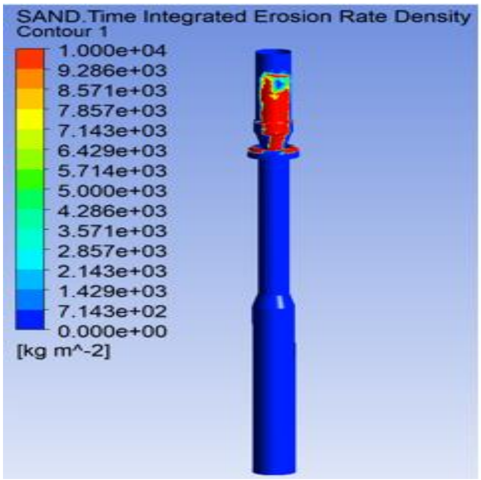

Figure 8: User defined erosion values at 15% opening and 2.5 kg/s mass flow for entire geometry.

The simulations were also carried out at different flows rate and when comparing the simulated maximum erosion rates, it is seen that the highest value appears at the needle with 15% opening and 17,5 kg/s mass flow rate. This is also the scenario where the highest pressure drop and velocity appears. Overall, the needle and seat are subject to high erosion rate, especially at higher flow rates. Erosion in the downstream (DS) region seems to depend on the velocity profile. At 50% opening the velocity jet profile is wide and reaches far DS. The erosion is spread on a large portion of the pipe walls. When the opening decreases, the velocity jet profile gets more focused, leading to higher erosion rate at a smaller area of the pipe wall. At 15% the jet is so focused that the particles are carried out of the model nearly without impacting the walls. An example of the observed erosion rates at 15% open for different flow rates is given in Table 1.

Table 1: Comparing erosion rates at 15% opening.

		Max erosion simulated (kg/m^2)		
Opening	**Mass flow rate kg/s**	**DS**	**Needle (10^8)**	**Seat (10^8)**
15 %	2.5	4382	6.74	0.535
15 %	7.5	0	90.6	9.37
15 %	12.5	0	180	11.8
15 %	17.5	306161	835	35.8

4. Summary and Discussions

The carried-out simulations have in the study shown that pressure drop affects the velocity jet shape and impact region. In the 50% opening case, the pressure drop creates a wide jet stream region that, in turn, will cause an increased wall impact region. When the opening decreases, to 35% and 25%, the jet stream gets more focused. The velocity jet impacts a smaller area of the pipe wall, which in turn creates possibilities for increased erosion rate. The high pressure drop in the 15% opening case creates a high-focused jet stream in the middle of the DS pipe, leading to low wall interaction. Overall, the 50% valve opening shows the lowest pressure drops and velocities. This is the case with the largest erosion regions, but the lowest erosion rates. The erosion is evenly distributed between the three critical parts, relative to the other opening cases.

Acknowledgment

We would like to thank Eric Stephen, Kristian Ringstad, Jon Ramstad for their contribution.

References

Bishnoi, P., et al. (2021). Effect of Velocity Variation on the Rate of Erosion Wear at the Elbow of the Slurry Pipe by Using CFD. Advances in Fluid and Thermal Engineering, Springer**:** 305-312.

Desale, G. R., et al. (2009). "Particle size effects on the slurry erosion of aluminium alloy (AA 6063)." Wear **266**(11-12): 1066-1071.

DNV, G. (2015). "Recommended Practice–DNVGL-RP-O501–Managing Sand Production and Erosion." DNV GL: Akershus, Norway: 3-60.

Feng, Z. and A. Ball (1999). "The erosion of four materials using seven erodents—towards an understanding." Wear **233**: 674-684.

Haugen, K., et al. (1995). "Sand erosion of wear-resistant materials: Erosion in choke valves." Wear **186**: 179-188.

Mathew, A. P. (2017). "Erosion rate prediction in single and multiphase flow using CFD." Int. Res. J. Eng. Technol **4**(7): 766-769.

McLaury, B. S., et al. (2000). "Effect of entrance shape on erosion in the throat of chokes." J. Energy Resour. Technol. **122**(4): 198-204.

Oka, Y. I., et al. (2005). "Practical estimation of erosion damage caused by solid particle impact: Part 1: Effects of impact parameters on a predictive equation." Wear **259**(1-6): 95-101.

Raghavendra, H., et al. (2014). "Simulation of erosion wear in choke valves using CFD." International Journal of Engineering Research & Technology **3**(7).

Wallace, M., et al. (2004). "Prediction of impact erosion in valve geometries." Wear **256**(9-10): 927-936.

Wheeler, D., et al. (2006). "Application of diamond to enhance choke valve life in erosive duties." Wear **261**(10): 1087-1094.