CFD STUDY TO RESOLVE CORROSION PROBLEMS IN REFINERY PROCESS UNITS

FIRMANSYAH, T., SULEIMAN, M. I., RAKIB, M. A., AND AL MUSHARFY, M.

ADNOC Refining Research Centre Division (RCD)
Abu Dhabi National Oil Company (ADNOC) Refining
PO Box 3593, Abu Dhabi, UAE.
e-mail: tfirmansyah@adnoc.ae

Key words: Computational Fluid Dynamics, Corrosion, Heat Exchangers

Summary. One of the chronic corrosion problems reported is the premature failure of reactor effluent exchangers of hydrotreating, and hydrocracking units; in particular the stripper feed and bottom product streams exchangers. The nature of the corrosion has been always severe thinning of the tubes outer surface and sometimes localized pitting and perforations. Corrosion is usually observed in one specific zone of the exchanger tube bundle, viz. areas between 4 to 8 o'clock shell outlet side, downstream of the last shell baffle and before the tube sheet. This paper presents one case corrosion history investigation using CFD simulations to study flow patterns inside the shell side of exchanger. The results have indicated that the corrosion inherent area had near zero velocity resulting in fluid stagnation. The low velocity allows the suspended solids and salt-saturated water droplet to settle and form a thin layer of hard deposit under which severe corrosion ensues. By changing fluid velocity and baffle opening orientation the CFD simulation was able to indicate the best condition for minimizing stagnation and therefore mitigating the under deposit corrosion.

1 INTRODUCTION

Use of Computational Fluid Dynamics (CFD) simulations as engineering analysis tool in fluid flow and thermal distribution study has recently gained big attention in many industries including oil and gas. Many CFD studies on the impacts of flow and thermal distribution to the occurrence of erosion and corrosion have already been published [1-5]. Most notable Zhu et al.[1] discussed the application of CFD to investigate the crystal and deposition behavior of ammonium chloride using multiphase flow simulation. Results from their CFD study showed distribution of NH4Cl salt granules at the inlet of reactor effluent air cooler tube and most importantly, 3-D visual mapping of concentration of solids deposition, which in turn indicated the location of under deposit corrosion. So, CFD has shown its benefit and advantage as troubleshooting analysis tool.

This paper discusses the premature failure of reactor effluent exchangers of hydrotreating, and hydrocracking units; in particular the stripper feed and bottom product streams exchangers. The tube bundle is U-type and metallurgy of tube is carbon steel conforming to ASTM A-179. The failure is due to external corrosion (from the shell side) of the tube caused by under deposit corrosion. It is concentrated at the bottom rows of the tubes near the channel side tube sheet around 6 o'clock position of the exchanger shell and downstream of the last baffle before the tube sheet. Leak was noticed in 30 nos. of tubes. A photograph depicting a sample of failed tubes is shown in Figure 1.

Figure 1: Photograph of the failed tubes

As part of troubleshooting strategies, CFD was employed to study the flow patterns across the shell side of the Heat Exchanger, in particular near the region where the corrosion occurred. CFD showed full insights of flow pattern inside the shell side of the Heat Exchanger such as regions of stagnant, i.e. close to zero velocities that may induce solids deposition and thus under deposit corrosion.

The CFD model covers the shell geometries including all the internal baffles and also the impingement plate downstream of the inlet, as shown in Figure 2.

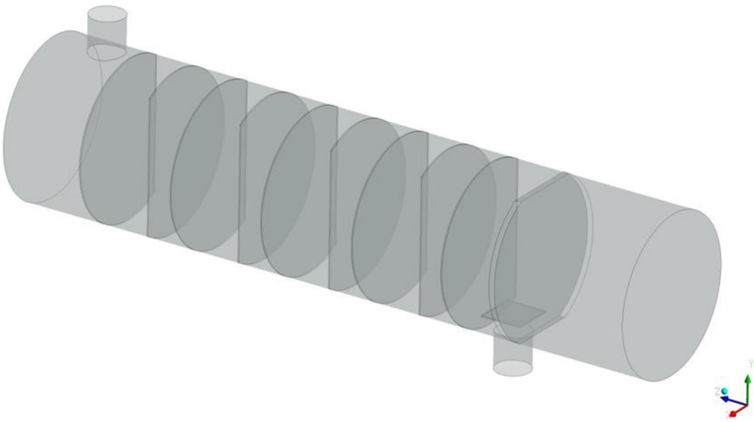

Figure 2: CFD geometry of heat exchanger

2 NUMERICAL MODEL AND RESULTS

Initially, baseline simulation was carried out using exactly the existing condition of baffle arrangement with the normal operating flow, i.e. 60% of design flow rate. The design flow rate was 93 kg/s. After evaluating the results of baseline simulations further four CFD simulations were performed with modifications of last baffle geometry and flow rate. Table 1 lists all the CFD simulation cases completed for this study.

No	Case	Geometry	Flow Rate (% of Design Flow)
1	Baseline	Existing	60
2	Modification #1	Last Baffle with One Hole	60
3	Modification #2	Last Baffle with Four Holes	60
4	Modification #3	Last Baffle with Full Bottom Cut-out	60
5	Modification #4	Last Baffle with Full Bottom Cut-out	100

Table 1: Simulation Cases

Figure 3 shows the resulting velocity contours of baseline simulation with red color representing regions with velocity of 0.1 m/s and above and dark blue regions with velocity close to zero, i.e. stagnant regions. The velocity contours are displayed in four different planes with three different levels of horizontal planes and one vertical plane to visually aid the spatial distribution of velocities. Clearly big stagnant zones appeared at the right side of the fluid inlet, which is expected since it is enclosed by the tube plate and majority of the fluid flows in the opposite direction towards the outlet. Another dominant stagnant zone was near the base of the shell opposite of the outlet. This is the area where under deposit corrosion occurred and led to the tubes leakage.

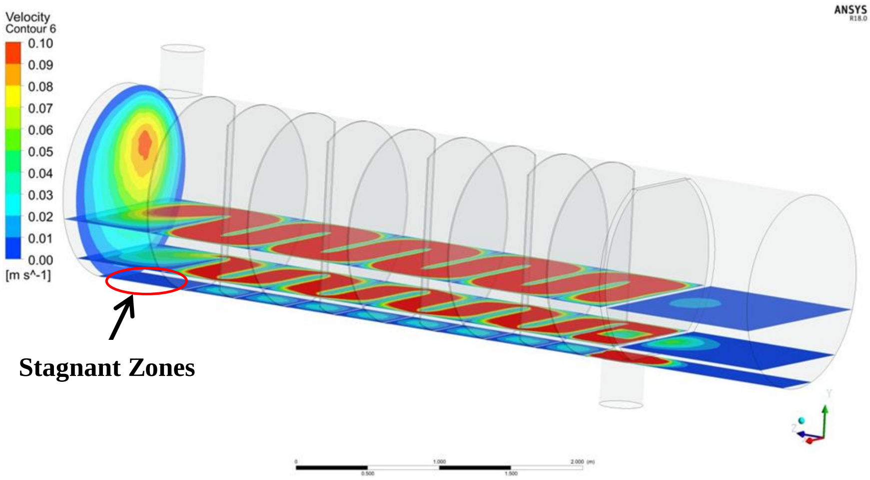

Figure 3: Resulting velocity contours of the existing geometry and operation condition

The CFD results have thus indicated that the corrosion inherent area had near zero velocity resulting in fluid stagnation. The low velocity allows the suspended solids and salt-saturated water droplet to settle and form a thin layer of hard deposit under which severe corrosion ensues.

Four proposed modifications in terms of baffle geometry and flow rate were conducted to help remove this area of stagnation. Modification #4 is the most optimum modification in terms of velocity increase to remove the areas of solids deposition, as depicted in Figure 4.

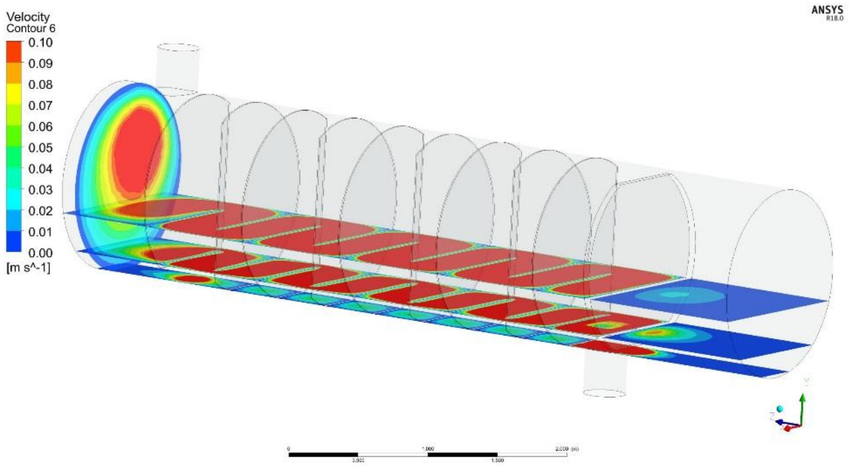

Figure 4: Resulting velocity contours of the modified geometry and operation condition

3 CONCLUSIONS

Computational Fluid Dynamics (CFD) was employed in this study to look into the underlying flow patterns across the shell side of the Heat Exchanger, in particular near the region where the corrosion occurred.

Four CFD simulation cases were performed including the existing geometry of the shell side of the Heat Exchanger and three modifications involving changes in baffle geometry and fluid flow rates. Results from CFD showed that predominant stagnant zone was present near the base of the original shell, opposite of the outlet where the tubes leakage occurred. This indicated that the corrosion inherent area had near zero velocity resulting in fluid stagnation. The low velocity allows the suspended solids and salt-saturated water droplet to settle and form a thin layer of hard deposit under which severe corrosion ensues.

By changing fluid velocity and baffle opening orientation the CFD simulation was able to indicate the best condition for minimizing stagnation and therefore mitigating the under deposit corrosion.

REFERENCES

[1] M. Zhu, G. Ou, H. Jin, K. Wang, Z. Zheng, ”Top of the REAC Tube Corrosion Induced by Under Deposit Corrosion of Ammonium Chloride and Erosion Corrosion,” Engineering Failure Analysis, 57 (2015), pp. 483-489.

[2] S. Nesic, “Using Computational Fluid Dynamics in Combating Erosion-Corrosion,” Chemical Engineering Science, 61 (2006), pp. 4086-4097.

[3] X. Chen, B.S. McLaury, S.A. Shirazi, “Application and Experimental Validation of a Computational Fluid Dynamics (CFD)-based Erosion Prediction Model in Elbows and Plugged Tees,” Computers & Fluids, 33 (2004), pp. 1251-1272.

[4] M.H. Wang, C. Huang, K. Nandakumar, P. Minev, J. Luo, S. Chiovelli, “Computational Fluid Dynamics Modeling and Experimental Study of Erosion in Slurry Jet Flows,” International Journal of Computational Fluid Dynamics, 23 (2009), pp. 155-172.

[5] A. Jatale, M. Srinivasa, “CFD Modeling of Fouling in Crude Oil Refinery Heat Exchangers,” AIChe Spring Meeting, 11th Global Congress on Process Safety, 2015.