Numerical study of the fouling effect on wet cooling towers designed to CSP plants

Afaf Zaza\textsuperscript{a,b,*}, Nour Eddine Laadel\textsuperscript{a}, El Ghali Bennouna\textsuperscript{a}, Youness El Hammami\textsuperscript{b}, Mourad Taha Janan\textsuperscript{c}

\textsuperscript{a}Thermal Systems Department, Institut de Recherche en Énergie Solaire et Énergies Nouvelles-IRESEN, 16, rue Amir Sidi Mohamed Souissi, Rabat, Morocco
\textsuperscript{b}Laboratory of Mechanics, Process, Energy, and Environment laboratoire de mécanique de procédé de l’enregie et de l’environement., Ecole nationale des ciences appliquén ENSA, Université ibno Zohr, Avenue Tamasloult, 80000, Agadir, Morocco
\textsuperscript{c}laboratoire de Mécanique Appliquée et Technologies, Centre de Recherche en Sciences et Technologies Industrielles et de la Santé, École Normale Supérieure de l’Enseignement Technique, ENSET, Mohammed V University in Rabat, Avenue de l’armée royal, RB6207, Rabat-

Abstract

As fouling issues have been gaining more and more attention every day, understanding the fouling behaviour on heat exchangers is the main way to increase knowledge on fouling effect, then reducing the waste heat on an industrial process. Cooling towers are widely used in CSP plants for energy transfers and utilization. However, air-side and water-side are usually unclean. The mineral salts contained in cooling water are deposited on the surface of heat exchangers, which reduce the heat transfer performance significantly and threatens the operating stability of the system. Meanwhile, the additional economic losses are caused due to the failure of heat exchange tubes and the soot-blowing process. Therefore, fouling is a major issue for heat exchangers and it should be fully taken into account in the heat exchanger design process. In this purpose, an apparatus scale unit of cooling towers designed for CSP plant was installed in the Green Energy Park (GEP) research platform, located in Benguerir, Morocco, in the framework of the MinwaterCSP project. In this study, we investigated the effects of fouling in cooling towers numerically for cross-flow heat exchangers tube geometry. The equation system governing the problem has been based on the finite volume method. The studied flow considered turbulent and modeled by the K-\(\epsilon\) standard model, known to generate satisfactory results for this type of flow. A mesh sensitivity study was also carried out to choose an optimal mesh. The obtained results show that the heat transfer efficiency with the polymer is greater than with galvanized steel in the absence of fouling. In case of dirty deposits, the efficiency drops for both materials, with the increase of the resistance of the fouling. For the polymer, the efficiency of the heat exchange decreases by up 4% and 3% for galvanized steel. Moreover, the fouling resistance is more higher on galvanized compared to polymer. This behaviour is due to the wall surface temperature of the two tubes, which are higher in the polymer than steel, which justified the rapid rate of deposition of the mass. The proposed model is validated by...

* Corresponding author: Nahla Bouaziz Tel.: +21697452626;
E-mail address: zaza@iresen.org

1876-6102 © 2019 The Authors. Published by Elsevier Ltd.
This is an open access article under the CC BY-NC-ND license (https://creativecommons.org/licenses/by-nc-nd/4.0/)
Selection and peer-review under responsibility of the scientific committee of Technologies and Materials for Renewable Energy, Environment and Sustainability, TMREES18.
10.1016/j.egypro.2018.11.289
Abstract

As fouling issues have been gaining more and more attention everyday, understanding the fouling behaviour on heat exchangers is the main way to increase knowledge on fouling effect, then reducing the waste heat on an industrial process. Cooling towers are widely used in CSP plants for energy transfers and utilization. However, air-side and water-side are usually unclean. The results show that the heat transfer efficiency with the polymer is greater than with galvanized steel in the absence of fouling.

1. Introduction

The fouling described as the unwanted particles that occurs on heat surfaces, it’s a severe problem that decreased by time the equipment performances[1]. In CSP plants process, fouling take place on heat exchangers inside the cooling tower tubes bundles. Deterioration of the heat exchanger surface leads to the decreasing on the process performances, moreover, a damaging effect was counted on the cleaning cost, design and environment [2,3].

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\rho)</td>
<td>Density</td>
</tr>
<tr>
<td>t</td>
<td>Time</td>
</tr>
<tr>
<td>(\nu)</td>
<td>Velocity</td>
</tr>
<tr>
<td>(S_m)</td>
<td>Term of mass source</td>
</tr>
<tr>
<td>P</td>
<td>Pressure</td>
</tr>
<tr>
<td>(\tau)</td>
<td>Cauchy constraint</td>
</tr>
<tr>
<td>F</td>
<td>Extern force</td>
</tr>
<tr>
<td>(\vec{g})</td>
<td>Gravity vector</td>
</tr>
<tr>
<td>(K_eff)</td>
<td>Conductivity effectiveness.</td>
</tr>
<tr>
<td>(\dot{j}_j)</td>
<td>Dissemination of cash flow j.</td>
</tr>
<tr>
<td>h</td>
<td>Enthalpy</td>
</tr>
<tr>
<td>(S_h)</td>
<td>Source</td>
</tr>
<tr>
<td>(E)</td>
<td>Specific energy</td>
</tr>
<tr>
<td>(G_k)</td>
<td>Generation of the turbulence of the kinetic energy due to the gradient of the average speed.</td>
</tr>
<tr>
<td>(G_B)</td>
<td>Generation of turbulence of kinetic energy due to buoyancy.</td>
</tr>
<tr>
<td>(Y_M)</td>
<td>Represents the contribution of the fluctuating dilatation in the compressible turbulence with respect to the rate of dissipation.</td>
</tr>
<tr>
<td>(\sigma_k)</td>
<td>Turbulent Prandtl numbers of k and e.</td>
</tr>
<tr>
<td>(m_d)</td>
<td>Mass deposition rate</td>
</tr>
<tr>
<td>(m_r)</td>
<td>Mass removal rate</td>
</tr>
<tr>
<td>(\beta)</td>
<td>Mass transfer coefficient</td>
</tr>
<tr>
<td>(C_B)</td>
<td>Concentration of the salts of the fluid in the shell.</td>
</tr>
<tr>
<td>(C_I)</td>
<td>Concentration of salts on the interface.</td>
</tr>
<tr>
<td>n</td>
<td>Order of the reaction</td>
</tr>
<tr>
<td>(K_0)</td>
<td>Pre-exponential Factor.</td>
</tr>
<tr>
<td>(Ea)</td>
<td>Activation energy.</td>
</tr>
<tr>
<td>R</td>
<td>Universelle Constante</td>
</tr>
<tr>
<td>Ti</td>
<td>Interface temperature.</td>
</tr>
<tr>
<td>(K_r)</td>
<td>Arrhenius Reaction rate coefficients in function of temperature.</td>
</tr>
<tr>
<td>(\rho)</td>
<td>Density of the fluid</td>
</tr>
<tr>
<td>(\mu)</td>
<td>Viscosity of the fluid</td>
</tr>
</tbody>
</table>
cooling water used in wet cooling contain insoluble salts that generate a supersaturation on the surface due to temperature differences.

The most famous salt that adheres to the surfaces by crystallization is calcium carbonate. On desalination plants under normal concentration, conditions for 1 million gallons/day a maximum of about 1400 kg of calcium carbonate or 800 kg magnesium could be precipitated each day. In terms of scale thickness, this possibly will characterize an accumulation of 0.1 mm per day[1]. CaCo₃ scale deposits on the tube wall of a heat exchanger, the first layer is formed on the surface while subsequent layers grow on CaCo₃ crystals. The crystallization mechanism arises depending on temperature and supersaturating degree and the crystal can form around a nucleus[1]. The mechanism of crystallization on surfaces is complex, and it is challenging to describe it in mathematical terms, although efforts have been made. The first researchers that opened the door and give their support to studying the fouling phenomena from the scientific viewpoint are Kern and Seaton and they developed the Kern- Seaton model to predict the fouling behavior on heat exchangers surface[4]. The approach is based on the mass deposition rate and removal one from the heat surface as mentioned in equation (1):

\[
\frac{dm}{dt} = m_d - m_r
\]  

Many scientists based on kern and Seaton crystallization model are presented in the literature[5–7], they refined the kern and Seaton model by introducing new parameters in order to elevate the fouling prediction behavior. As revealed by Watkinson and Epstein, they proposed a model based on the sticking probability and the mass transfer coefficient to calculate the mass deposition rate of the fouling[8]. Beside other investigators offered a general asymptotic fouling model that takes into account the driving force concept for the deposit accumulation [9]. The fouling is a complex mechanism because of the integration of chemical and physical parameters. Thus to improve knowledge about the flow properties of heat exchangers in complex geometries, refined techniques, such as Computational Fluid Dynamics (CFD) are used. CFD is based on finite volume method to solve the governing equations of the fluid flow and heat and mass transfer [7]. Current study advanced by Paakonen et al. using the CFD modeling to predict the CaCO₃ behavior on heat exchanger surface[10,11]. She predicts the mass deposition rate of the fouling on heat surfaces by taking in consideration the bulk fluid saturated concentration.

In this work, a numerical study had been investigated to predict the crystallization behavior of CaCo₃ deposits on two different surfaces. The study was conducted by Ansys fluent software for a fouling phenomenon occurred on wet cooling towers. The modeling based on finite volume methods and take in consideration the turbulence modeling in the vicinity of the tubes.

2. Mathematical model

2.1. Governing equations

The modeling is established with the commercial CFD software ANSYS FLUENT R18.0[12]. The governing equations for conservation of mass (Eq. (2)), momentum (Eq. (3)), and energy (Eq. (4)) are solved with the finite-difference interpolation technique employing a control-volume based on discretization method with a pressure-based solver algorithm[12].

- Continuity equation
  \[
  \frac{\partial \rho}{\partial t} + \nabla.(\rho \vec{v}) = S_m
  \]  

- Momentum equation
  \[
  \frac{d(\rho \vec{v})}{dt} = \nabla.(\rho \vec{v} \vec{v}) = -\nabla P + \nabla(\tau) + \rho \vec{g} + \vec{F}
  \]

Where \( \vec{\tau} \) is the Cauchy constraint.
With $\frac{2}{3} v^2 I$ indicating dilation.

- *Energy equation*

$$\frac{\partial \rho E}{\partial t} + \nabla \cdot (\rho (E - p)) = \nabla (K_{\text{eff}} \nabla T - \sum_i h_j j_i + (\tau_{\text{eff}} \cdot \nu)) + S_h$$

Where $E$ is the specific energy defined as follows $E = h - \frac{p}{\rho} + \frac{v^2}{2}$

### 2.2. Turbulence modeling

The flow studied had a turbulent flow with Reynolds number of 30,000, therefore a K- $\varepsilon$ model was used to model the turbulence flow. This model based on the resolution of two separate equation transport, one for the turbulent kinetic energy $k$ (Eq. (5)), and the second for the dissipation rate $\varepsilon$ (Eq. (6)).

$$\frac{d(\rho k)}{dt} + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_i} \left[ (\mu + \frac{\mu_l}{\sigma_k}) \frac{\partial k}{\partial x_i} \right] + G_k + G_b + \rho \varepsilon - Y_M + S_k$$

$$\frac{d(\rho \varepsilon)}{dt} + \frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_i} \left[ (\mu + \frac{\mu_l}{\sigma_\varepsilon}) \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1e} \frac{\varepsilon}{k} (G_k + C_{2e} G_b) + C_{3e} \rho \frac{\varepsilon^2}{k} + S_\varepsilon$$

$$\mu_l = \rho C_{\mu} \frac{k^2}{\varepsilon}$$

With:

- $G_k$: Generation of the turbulence of the kinetic energy due to the gradient of the average speed.
- $G_b$: Generation of turbulence of kinetic energy due to buoyancy.
- $Y_M$: Represents the contribution of the fluctuating dilatation in the compressible turbulence with respect to the rate of dissipation.

- $\sigma_k$, $\sigma_\varepsilon$: Turbulent Prandtl numbers of $k$ and $\varepsilon$.
- $C_{1e}$, $C_{2e}$, $C_{3e}$: Constants having the following values 1.44; 1.92; 0.09 respectively.

The mass transfer is modeled by the following equation:

$$\frac{\partial}{\partial t} (\rho Y_i) + \nabla (\rho \nabla Y_i) = -\nabla J_i + S_i$$

Where $Y_i$ represents the mass fraction of the species $i$, and $S_i$ the source term, which will be modeled by the fouling model using the user-predefined functions in Ansys Fluent (User defined functions).

### 2.3. Fouling model

The mass flux of ions in the vicinity of the surface is described by the relation below

$$m_d = \beta (C_b - C_i)$$
With $\beta$: Mass transfer coefficient $\beta = \frac{shD}{D_h}$ being diffusion coefficient $D_h$ is hydraulic diameter and $S_h$ the Sherwood number.

Below, the rate of deposition of the material due to the integration of the surface, which means the integration of the ions by the exchange surface:

$$m_d = K_r (C_b - C_{sat})^n$$

With $K_r$: Arrhenius Reaction rate coefficients in function of temperature.

$$K_r = K_0 e^{-Ea/RT}$$

The order $n$ is assumed to be 2 [13]. What makes the previous equation (9) 

$$m_d = K_r (C_b - C_{sat})^2 = K_0 e^{-Ea/RT} (C_b - C_{sat})^2$$

To take into account the low velocity, a mass transfer coefficient at low velocity should be integrated to the mass deposited equation. Therefore a reformulation of (Eq.9) and (Eq.10) reproduce the following equation:

$$m_d = \beta \left[ \frac{1}{2} \left( \frac{\beta}{K_r} \right) + (C_b - C_{sat}) - \frac{1}{4} \left( \frac{\beta}{K_r} \right)^2 + \frac{\beta}{K_r} (C_b - C_{sat}) \right]$$

From the experimental work of Tina.M. Pääkkönen[10,14] fouling by crystallization is mainly controlled by the integration of the surface. Thus, the residence time of the fluid in the vicinity of the surface which depends on the speed of friction, affects the crystallization. Therefore, the time scale factor, $t_{sf}$ of Epstein (Time scaling factor) must be included in the fouling model.

$$m_d = K_r' (C_b - C_{sat})^2 t_{sf}$$

$$t_{sf} = \frac{\mu_l}{\rho_l V^2}, \quad K_r' = K_0 e^{-Ea/RT}$$

Where $\rho_l$ and $\mu_l$ are respectively the density and the viscosity of the fluid.

The mass fraction of CaCO$_3$ that will be used as a limiting condition in the CFD model is obtained by multiplying the molar concentration by the molar mass divided by the density of the fluid.

The properties of the fluid (density, viscosity, diffusivity, saturation concentration) are calculated from the CFD model using the temperature-dependent equations.

The temperature $T_i$ is modeled in the CFD model, and the speed of friction is obtained from the following relation:

$$V = \sqrt{\frac{\tau_w}{\rho_l}}$$

where the shear stress $\tau_w$ is calculated by the CFD model.

Returning to the case of low flow speeds, as mentioned, the mass transfer occurs in the process of fouling, therefore it must be integrated in the model of fouling, and the latter becomes:

$$m_d = K_0 e^{-Ea/RT} (C_i - C_{sat})^2 \frac{\mu_l}{\rho_l V^2}$$

where $C_i$ is the concentration at the interface between the surface and the fluid. It is calculated by equation (8).

The thermal resistance of the fouling layer is modeled by the following equation:

$$R_f = \frac{m_d.t}{\rho_f \lambda_f}$$

$\rho_f$ and $\lambda_f$ are assumed to be constant based on the work of Pääkkönen et al.[10,13].
2.4. Model geometry

For perfect modeling the studied issue, the geometry of the exchanger heat should take a careful attention. For our case the geometry was set with Fluent (design modeler). The cold water inlet was in direct contact with the polymer and galvanized tubes.

A 2D model grid and geometry (Fig. 1) are created.

![Computational geometry with enlargement of the grid](image)

The dimensions of the model geometry agree to the experimental heat exchanger. Two-dimensional model is chosen to save computational time. A mesh sensitivity study was also carried out to choose an optimal mesh. The simulation was carried out with a quadrilateral mesh of 53 077 cells, in order to minimize the computation time.

In order to show the boundary layer that forms on the cylinder, it is necessary to have a too fine mesh in the vicinity of the wall (Fig.1).

For this type of mesh, it is necessary to determine its intended thickness so that the simulation generates good results in the vicinity of the wall of the cylinder.

2.5. Boundary conditions

To identify the physical boundaries of the geometry an appropriate boundary conditions are needed. The boundary values correspond to the operating conditions of the experimental test. At the inlet, the velocity transverse factor is equal to 1.77m/s for the cold water, and the inlet temperature is 291K. The CaCo3 bulk concentration is set at 390 mg/l. At the outlet, the cold-water temperature is fixed at 293K. The other boundaries are wall boundaries the condition was set as defined by FLUENT. The Reynolds number at the cold side is fixed at 30 000. The density, diffusivity, viscosity and saturation concentration of the hard water are calculated and taken into account on the model.

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Flat Values</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet cold water</td>
<td>V=Vin</td>
<td>1.77 m/s</td>
</tr>
<tr>
<td></td>
<td>T=Tin</td>
<td>291K</td>
</tr>
<tr>
<td></td>
<td>C=Cb</td>
<td>390 mg/l</td>
</tr>
</tbody>
</table>
Table 1: Specifications of the considered tubes.

<table>
<thead>
<tr>
<th>Specification</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outlet cold water T</td>
<td>293 K</td>
</tr>
<tr>
<td>Heated wall Qpoly</td>
<td>6096 W/m</td>
</tr>
<tr>
<td>Qgalva</td>
<td>5134 W/m</td>
</tr>
</tbody>
</table>

3. Model validation

Checking the results obtained accuracy is the most important point to validate the model, to do so is to compare the simulated result with the experimental data. For this purpose, the adopted numerical model CFD was validated using the experimental results of the fouling resistance for galvanized steel (Figure 2.b) and the polymer (Figure 2.a).

The experimental data was calculated based on the total resistance which calculated by the flow equation

\[ R = \frac{1}{UA_i} - \frac{1}{UA_f} \]

where the U is the overall heat transfer coefficient and A is the heated tube surface.

![Figure 2: CFD and experimental results for galvanized (a) and polymer tubes (b)](image)

4. Simulation results and discussion

After the simulations were over, results were extracted in the form of contour plots, vector plots and XY plots. For the XY plots, the data sets were extracted along the lines as shown in Fig 1.

4.1. X Velocity

Fig 3 shows the velocity contour plots distribution around the heated tubes and Fig 4 presents a magnification of velocity contours of the two top tubes. The dark-red color regions indicate higher velocities and dark-blue color regions indicate reversed flow. Yellow and green indicate intermediate velocities. Light blue represents very low velocities. As results, the higher velocity was achieved at the left and right sides for the top tubes while low velocities are reached at the top and the bottom of the tubes. The low velocity behaviour can be explained by flow separation around the tubes. As the fluid approaches the tubes wall it is forced to separate and go around the tubes wall this flow separation causes low velocities at the bottom of the tubes[15]. The tubes position also could affect the velocity distribution, as we mentioned that the velocity is higher for the two top polymer and galvanized tubes then the velocity decreases gradually from the inlet to the outlet-studied domain.

Fig 5 shows the x velocity plot for the lines of data extraction shown in Fig 1. The gaps between the graphs curves are due to the presence of the tubes surfaces. The velocity at the forth lines is initially low, then it rises as the...
Model validation

Checking the results obtained accuracy is the most important point to validate the model, to do so is to compare the simulated result with the experimental data. For this purpose, the adopted numerical model CFD was validated using the experimental results of the fouling resistance for galvanized steel (Figure 2.b) and the polymer (Figure 2.a). The experimental data was calculated based on the total resistance which calculated by the flow equation

\[ R_{UA} = \frac{1}{U} - \frac{A}{U} \]

where the \( U \) is the overall heat transfer coefficient and \( A \) is the heated tube surface.

4. Simulation results and discussion

After the simulations were over, results were extracted in the form of contour plots, vector plots and XY plots. For the XY plots, the data sets were extracted along the lines as shown in Fig 1.

4.1. X Velocity

Fig 3 shows the velocity contour plots distribution around the heated tubes and Fig 4 presents a magnification of velocity contours of the two top tubes. The dark-red color regions indicates higher velocities and dark-blue color regions indicate reversed flow. Yellow and green indicate intermediate velocities. Light blue represents very low velocities. As results, the higher velocity was achieved at the left and right sides for the top tubes while low velocities are reached at the top and the bottom of the tubes. The low velocity behaviour can be explained by flow separation around the tubes. As the fluid approaches the tubes wall it is forced to separate and go around the tubes wall this flow separation causes low velocities at the bottom of the tubes [15]. The tubes position also could affect the velocity distribution, as we mentioned that the velocity is higher for the two top polymer and galvanized tubes then the velocity decreases gradually from the inlet to the outlet-studied domain.

Fig 5 shows the x velocity plot for the lines of data extraction shown in Fig. 1. The gaps between the graphs curves are due to the presence of the tubes surfaces. The velocity at the forth lines is initially low, then it rises as the fluid approaches the tubes wall. Then it falls again as it enters the tubes region. Between the tubes, the velocity reached the higher value around 2.7 m/s. The predicted values give a good agreement with experimental results. At low velocity, the deposits fouling are more significant [16,17], which is proved experimentally in our case.

Figure 3: contour velocity simulation

Figure 4: Zoom of the contours (a) and vectors (b) velocity for the first top tubes.
Masse deposition rate
The fouling resistance was calculated based on the surface temperature using the followed equation

$$R = \frac{T_{wi} - T_{wf}}{Q}$$

(20)

Based on equations 14 and 15 the mass deposition layer was calculated as follows:

$$m_j = \frac{\rho_i(T_{wi} - T_{wf})A_l}{t.Q}$$

(21)

Fig.6 presents the mass deposition on both top tubes, the mass deposition increases with increasing the wall surface temperatures. The results show a similar mass deposition on different heated tubes verses wall temperatures, which is explained by the non-integration of the removal rate on the modeling equation, especially in polymer tube. Because experimentally, we mentioned the easy cleaning of the fouling layer from the polymer tubes.

4.2. Fouling resistance.

The fouling resistance calculated from (Eq.20) was presented at Fig.7. The results mentioned that the fouling resistance of the polymers is slightly lower than the galvanized one, which is adequate with the experimental test[18]. This behavior is due to the smooth surface, and the low roughness of polymer tubes compared with galvanized ones[17].
The fouling resistance was calculated based on the surface temperature using the followed equation:

$$wi - wf \times T_{TR} \times Q = 0$$  \tag{20}$$

Based on equations 14 and 15, the mass deposition layer was calculated as follows:

$$dm = \frac{\rho}{\lambda} \times t \times Q$$  \tag{21}$$

Fig. 6 presents the mass deposition on both top tubes, the mass deposition increases with increasing the wall surface temperatures. The results show a similar mass deposition on different heated tubes versus wall temperatures, which is explained by the non-integration of the removal rate on the modeling equation, especially in polymer tubes. Because experimentally, we mentioned the easy cleaning of the fouling layer from the polymer tubes.

4.2. Fouling resistance.

The fouling resistance calculated from (Eq. 20) was presented in Fig. 7. The results mentioned that the fouling resistance of the polymers is slightly lower than the galvanized one, which is adequate with the experimental test\[18\]. This behavior is due to the smooth surface, and the low roughness of polymer tubes compared with galvanized ones\[17\].

Conclusion

The fouling model still a complicated phenomenon. To predict perfectly a fouling model, the forces of the particle tracking as well is still awaited. The simulation of the fouling crystallization includes the wall surfaces of the tubes bundles. The simulation results showed a good agreement with experimental data and followed the trend of the other calculations. Surface materials were closely investigated. The results showed that the mass deposition on polymer tubes is similar to the galvanized ones due to the non-implementation of the removal rate for polymers deposit, because experimentally, we can easily clean and remove the fouling deposits from polymer surface than the galvanized steel one. Moreover, the fouling resistance is more significant on galvanized tubes than the polymer ones due to the higher roughness of the galvanized tubes. The modelling results are satisfactory enough to give an initial idea about the fouling behavior on heated tubes due to wet cooling which will be dealt with further works.

Figure 6: mass deposition rates on polymer and galvanized tubes

Figure 7: fouling resistance on both polymer and galvanized tubes
References