Memo

Stantec

RECEIVED
FEB 05 2013
DEPT OF ECOLOGY

To:	Mick Reeves Chief Engineer, Seattle Steam	From:	Fangbiao Lin, Ph.D. Principal, Applied Hydraulics
File:	181300289	Date:	February 5, 2013

Reference: CFD Modeling of Seattle Steam Effluent Mixing Zone

This memo summarizes the modeling approach and results of a computational fluid dynamics (CFD) study for delineating the effluent mixing zone for the Seattle Steam.

Project Background

Seattle Steam Company discharges backwash water from the ion exchange unit to Elliot Bay under Pier 57. Washington State Department of Ecology (DOE) issued a permit for this facility (WA-000150-3), which authorizes Seattle Steam to discharge softening regenerate water and storm water to Elliot Bay through the existing outfall subject to meet the stipulated conditions regarding flow, temperature, pH, and oil and grease. The following are two temperature conditions: 1) the temperature limit (16 °C) applies at the boundary of the defined mixing zone at 200 feet from the discharge point into Elliott Bay, and 2) when ambient water temperature exceeds 16°C, no temperature increase greater than 0.3°C will be allowed.

The Fact Sheet for NPDES Permit WA0001503 prepared by Ms. Jeanne Tran on April 17, 2012 requires performing a temperature mixing zone study to evaluate whether the effluent has a reasonable potential to cause a violation of the water quality temperature standards. To meet this permit condition this study was performed to delineate the mixing zone using the state-of-the-art CFD modeling approach. The modeling results demonstrate that the effluent mixing zone fully complies with the permit requirements for the temperature limit at the edge of the mixing zone.

CFD Modeling Approach

DOE accepts Visual Plume and CORMIX program for defining mixing zones from effluent discharges. Based on the communication with Anise Ahmed (DOE's technical expert) and Jeanne Tran, a three dimensional CFD model is also acceptable to delineate the detailed mixing zone, which could provide more details of the thermal mixing zone than CORMIX and Visual PLUMES. The CFD modeling solves a full set of the Reynolds-averaged Navier-Stokes equation, k-ε turbulence equations, and energy equations. It also models the buoyance effects resulting from the temperature difference between the effluent and seawater.

The CFD modeling approach was successfully used for predicting thermal mixing zones in Tennessee Valley Authority (TVA)'s Brown Ferry Nuclear Plant, as detailed in Lin (2003). The CFD results were fully verified by both laboratory and filed measurements, which demonstrated that this modeling approach can reliably predict thermal mixing zones (Lin, 2003). The CFD modeling was also successfully used for delineating effluent mixing zones for the outfalls of two wastewater treatment plants. Comparison of CFD results to the field dye testing also demonstrates that the CFD results match with field measurements (Lin, 2013).

Figure 1 shows the CFD model that was developed using a CFD software named ANSYS FLUENT (Version 14.0). FLUENT is commercially available CFD software and has been successfully used for modeling a variety of engineering flows. This model includes the 4 inch diameter outfall, a rectangular area of 600 ft x 300 ft, and 9 ft water depth. A field survey was performed on September 20, 2012 by Muir Contracting, Inc. to provide measurement of the geometrical dimensions of the outfall. The Finding Report of the field survey is included as an attachment of this memo. The CFD model was developed based on the dimensions from the field survey. A 9 ft water depth was modeled throughout the domain, which is conservative because the water becomes significantly deeper approximately ten feet from the sea wall. The assumption of a smaller water depth produces a larger mixing zone because less water is available for mixing. A total of 2.5 million elements of meshes were used to delineate the mixing zone with finer meshes clustered around the outfall and in the 200 ft mixing zone.

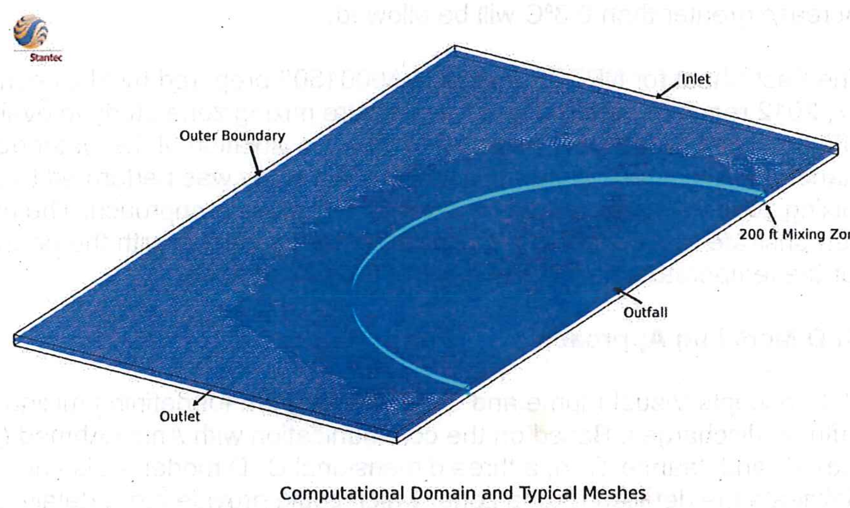

Figure 1: CFD Model Domain and Typical Meshes

The charts in Tide Currents of Puget Sound by Starpath (2009) indicate that the current is about 0.1 knot from the North to South. The direction of the current can be reversed and does not impact the size of the mixing zone. For example, if the current flows from South to the North, the plume just flips with its size unchanged. Three scenarios were modeled and their operating conditions were summarized in Table 1 below. Case No. 1 represents a normal summer seawater temperature and Case No. 2 represents a scenario with background temperature higher than 16 °C. Case No. 3 reduces the current speed by half to 0.05 knot and serves as a sensitivity study.

Table 1: Operating Conditions of the Two Modeled Scenarios

Case No.	Ambient Current (ft/s)	Ambient Temp (°C)	Seawater Density (kg/m^3)	Effluent Flow Rate (gpm)	Effluent Temp (°C)	Effluent Density (kg/m^3)
1	0.169	15.6	1016.2	50	29.5	996.0
2	0.169	16.5	1016.0	50	29.5	996.0
3	0.085	16.5	1016.0	50	29.5	996.0

These operating conditions are considered to be conservative, as explained below:

- The effluent temperature of 29.5 °C is the maximum effluent temperature (see Table 3 of Fact Sheet For NPDES Permit WA0001503)
- The effluent flow rate of 50 gpm is the pump flow rate, which amounts to 72,000 gpd that is 22,000 pgd higher than the permitted discharge of 50,000 gpd.
- Constant sea water density was used, which is considered to be conservative because denser water on upper layer will keep the warm discharge close to water surface and produce a larger mixing zone. The current seawater density was calculated based on the salinity of 22,400 TDS (or ppm), which is typical for Elliot Bay.

The CFD model does not model the wind impact because the outfall is located under the Pier 57 and it is expected that wind impact is not significant. In addition, the CFD model is a steady-state model that does not model the tide buildup. The effluent was treated as a conservative pollutant and the temperature was predicted by solving the flow, turbulence and energy equations. The mass diffusivity of 2.88 x 10^{-5} m^2/s was used in the model; therefore, diffusion between effluent and seawater is not significant.

Five types of boundary conditions were used in the CFD model, as shown in Figure 1 and detailed below:

- The current velocity was specified at the North model inlet.
- A velocity of 1.2792 ft/s was specified at the effluent outfall, which basically specifies an effluent flow of 50 gpm for the 4" diameter outfall.
- The water surface was treated as a flat surface that has a zero shear stress and allows water to freely move on the surface without any vertical velocity component.
- The sea bottom and seawall were treated as non-slip walls with zero velocity.
- The outer boundary of the model was specified as a symmetry boundary, allowing water to freely move on this boundary.

Results and Discussion

The temperature mixing zone of the Case 1 is shown in Figures 2 and 3. Figure 2 shows temperature contours at a 0.05 °C interval increasing from the background temperature (15.6 °C) to 0.5 °C above the background temperature (16.10 °C). Temperature higher than 16.05 °C will be in red color. Two planes through outfall center – X=0 and Y=0 planes -- are enlarged for better view. From this figure it can be seen that the thermal plume stays near the surface and the temperature at the edge of the 200 ft mixing zone is about 0.1 °C above the background temperature and the temperature is 0.3°C less than the 16 °C temperature limit. This figure also shows that warm water travels a bit upstream.

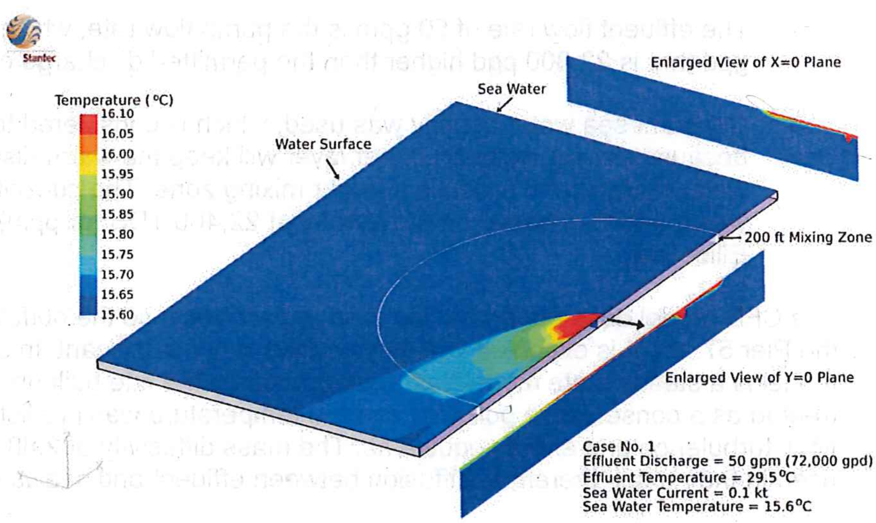

Figure 2: Temperature Distribution on Seawater Surface and Outfall Central Planes (Case 2)

Figure 3 plots the temperature distribution on the 200 ft mixing zone and three temperature iso-surfaces for 0.1, 0.2 and 0.3 °C above the back ground temperature. This figure shows that only a small area on the edge of the mixing zone experiences temperature increase and the maximum temperature is 15.71 °C. This figure also shows that the area of 15.7 °C (0.1 °C) extends slightly beyond the mixing zone, but areas where temperature is increased by 0.2 and 0.3 °C are relatively small.

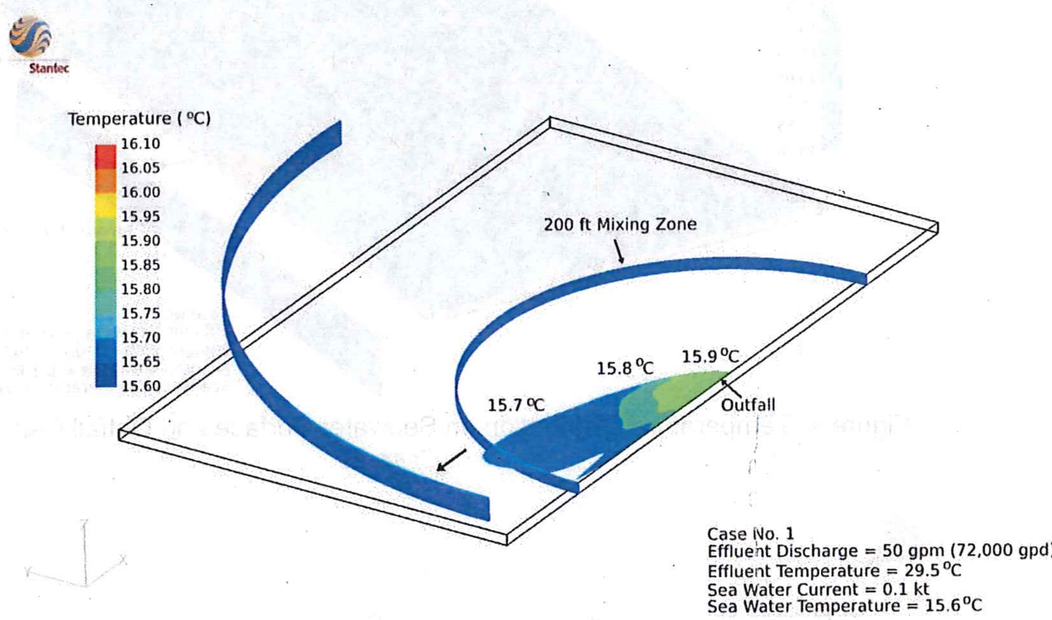

Figure 3: Temperature Distribution on Mixing Zone Edge and Thermal Plume (Case 1)

For Case No. 2, the CFD results are plotted in Figures 4 and 5. These figures are similar to Figures 2 and 3, and show that a temperature increase of less than 0.1 °C at the edge of the 200 ft mixing zone, which is much less than the 0.3 °C permitted.

For Case No. 3 with the current speed reduced by half (0.05 knot), the CFD results are plotted in Figures 6 and 7. These figures are similar to Figures 4 and 5, and also show that a temperature increases of less than 0.1 °C at the edge of the 200 ft mixing zone. Comparison Figure 6 to Figure 4 and Figure 7 to Figure 5 show that when current speed reduces, the mixing zone becomes rounder, but the temperature increase is still much less than the 0.3°C temperature increase permitted. The results of the sensitivity run confirmed that even current is much lower, the temperature conditions will still be met at this site.

The above observations are confirmed by the field monitoring results of this outfall over past years that have recorded no temperature increase at the edge of 200 ft mixing zone.

Reference: CFD Modeling of Seattle Steam Effluent Mixing Zone

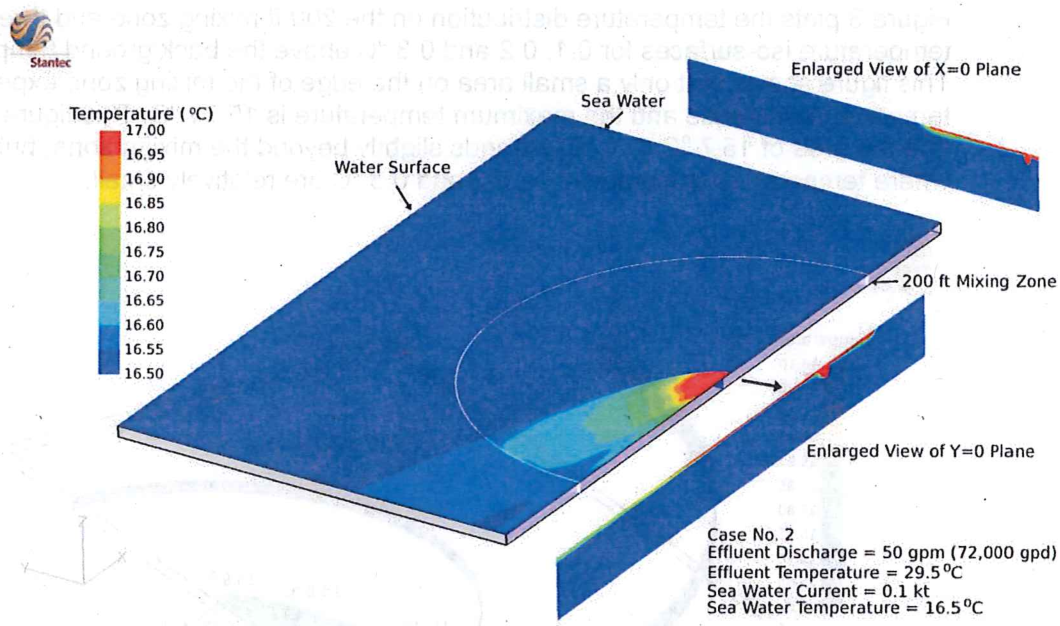

Figure 4: Temperature Distribution on Seawater Surface and Outfall Central Planes (Case 2)

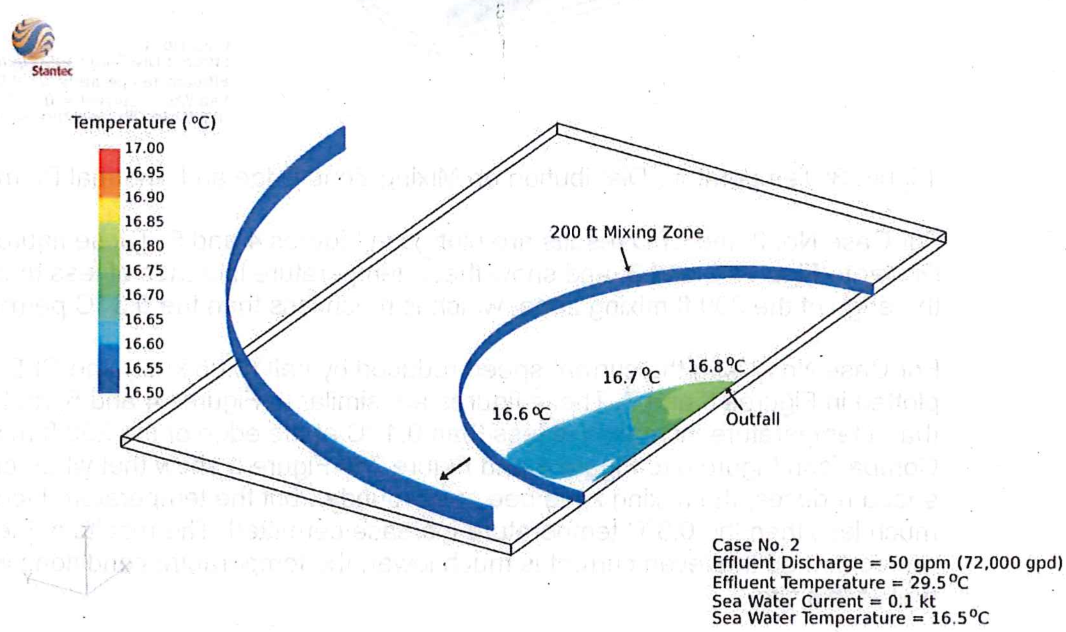

Figure 5: Temperature Distribution on Mixing Zone Edge and Thermal Plume (Case 2)

Reference: CFD Modeling of Seattle Steam Effluent Mixing Zone

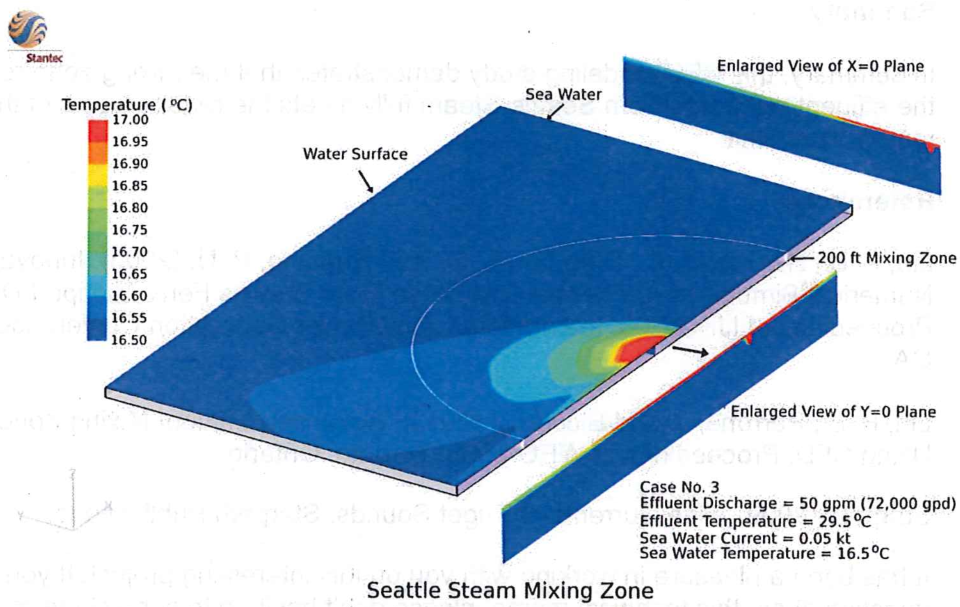

Figure 6: Temperature Distribution on Mixing Zone Edge and Thermal Plume (Case 3)

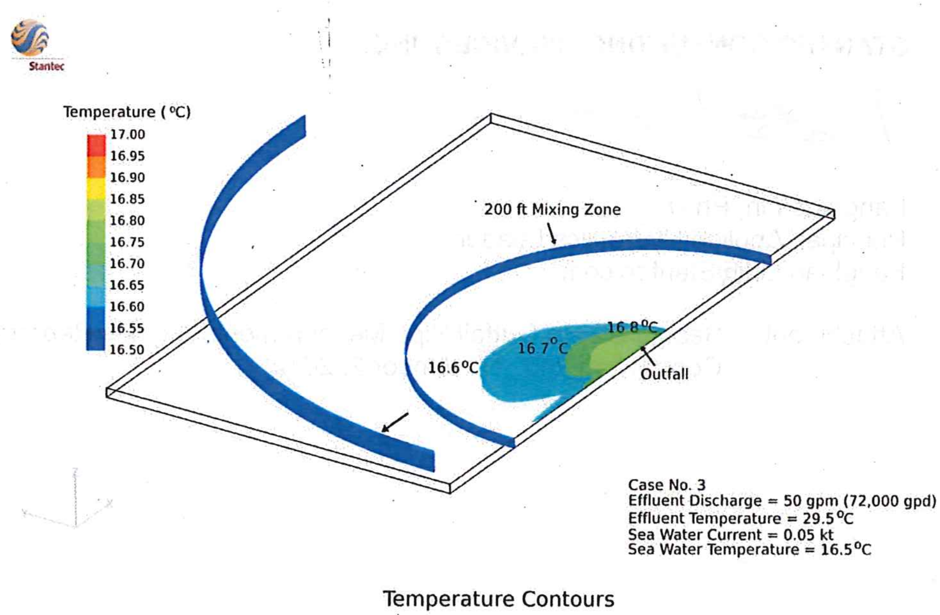

Figure 7: Temperature Distribution on Mixing Zone Edge and Thermal Plume (Case 3)

Summary

In summary, this CFD modeling study demonstrates that the mixing zone resulting from the effluent discharge from Seattle Steam fully meets the requirements of the permit on temperature limit.

References

Lin, F. B., Hecker, G. E., Brennan, T. S., and Hopping, P. N. (2003). Innovative 3-D Numerical Simulation of Thermal Discharge From Browns Ferry Multiport Diffusers. Proceedings of IJPGC'03: International Joint Power Generation Conference, Atlanta, GA.

Lin, F. B., Perrone, J. and Bicudo, J (2013). Advanced Effluent Mixing Zone Modeling Using CFD. Proceedings of WEO 2003, Toronto, Ontario.

Starpath (2009). Tide Currents of Puget Sounds. Starpath Publications.

It has been a pleasure in working with you on this interesting project. If you have any question about this technical memo, please don't hesitate to contact me at Fangbiao.Lin@stantec.com or (206) 459 6628.

STANTEC CONSULTING SERVICES, INC.

Fangbiao Lin, Ph.D.
Principal, Applied Hydraulics Leader
Fangbiao.Lin@Stantec.com

Attachment: Seattle Steam Outfall Pipe Measurement Findings Report (Muir Contracting, Inc., November 2, 2012)