COMPUTER MODELLING DISPERSION PLUME OF TEBAR - PETROBRÁS SUBMARINE OUTFALL

RICARDO FORTIS[1]; JAYME PINTO ORTIZ [2];
CLAUDIA CONDE LAMPARELLI[3]; and REGIS NIETO[4]

[1] Universidade de São Paulo, Escola Politécnica,
Departamento de Engenharia Hidráulica e Sanitária,
(Tel.: +55-11-3091-9669, e-mail: ricardo.fortis@poli.usp.br)
[2]Universidade de São Paulo, Escola Politécnica, Departamento de Engenharia Mecânica,
Av. Prof. Mello Moraes, 2231 CEP 05508-900, São Paulo, Brasil,
(Tel: +55-11-3091-5335, e-mail: jportiz@usp.br.)
Centro Universitário do Instituto Mauá de Tecnologia. São Caetano do Sul, SP, Brasil,
(e-mail: ortiz@maua.br.)
[3,4] Companhia de Tecnologia de Saneamento Ambiental – CETESB,
Av. Prof. Frederico Hermann Jr., 345 CEP 05459-900, São Paulo, Brasil
(Tel: +55-11-30306078, e-mail: clauidal@cetesb.sp.gov.br)

CFD is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations that govern these processes using a numerical algorithm (that is, on a computer). To apply CFD, the geometry of interest is first divided, or discretized, into a number of computational cells. Cells can have a variety of shapes. For 3-D problems, where the flow depends on all three spatial coordinates, hexahedral, tetrahedral, pyramidal, and prismatic shaped cells can be used.

Computer modeling use for the waste water dispersion process description in water bodies is a modern tool as a decision support for the environmental agencies and for the industries. A computer technique based in *CFD* through the finite volume technique, using the software *FLUENT,* which conduct to the dilution results of the plume in the near and far field is a powerful tool for the simulation of industrial waste dispersion process in a variable batimetries and considering discharges from multi-port diffusers with complex geometries (ORTIZ&BESSA, 2004). FLUENT can model the mixing and transport of chemical species by solving conservation equations describing convection, diffusion, and reaction sources for each component species.

The São Sebastião Channel (25km long, 40m depth and the narrowest section is 2km width) contains one of the largest oil terminals in Brazil (Tebar – Terminal Almirante Barroso of Petrobrás – Petróleo Brasileiro S.A.). The submarine outfalls are formed by two separated and independent outfall lines build of HDPE – High density polyethilene. Both of them has three risers with 0.15m of diameter, 1.5m of height and are located in a height between 19.15m and 25.45m. Were analyzed hydrographic and current data. The highest intensities of 0.90m/s (northward) and medium intensities between 0.40m/s and 0.60m/s (northward).

The effluent was sampled from April 2000 to March 2002 by Petrobrás. In this period, was founded high concentrations of ammonia (125.5mg/L in April 2001), trespassing the limited value for emission of CONAMA 20 Resolution – Art.21 (5mg/L). The salinity of effluent is in order of 52,8% at a temperature of 27.3°C for the same period. This range of

values results in a density of 1036.7 kg/m^3.

The geometry creation and mesh generation of São Sebastião Channel was based utilizing the software *Gambit*. The bathymetric data was taken from Nautical Map n°1643 from Marinha do Brasil. The correspondent dimensions used to create geometry and mesh generation are the follows: Length: 915m; Width: 525m; Maximum Height: 36m. The hexahdral structured mesh was built for the whole channel, including the contours around ports. The 3D mesh has a total of 655.452 volumes. The cells contain the following dimensions: length (Gx) and width (Gz) equal 2m, and height (Gy) of 4,33m.

For the simulations presented in this paper, was utilized a personal computer Pentium 4 – 1,5 GHz processor velocity and 1.0GB of RAM memory. Results of simulation by FLUENT are presented in the figures 1 and 2. The Fig. 1 shows concentration of ammonia at near-field along 50m of channel. The results present concentration value, C = 0,4mg/L (CONAMA Resolution), and represented by white color, is attended nearly in 20m far from discharge, but a strong impact benthic occurs because of the advection current action in the hydrodynamic process. The Fig. 2 shows longitudinal sections (along the six diffusers) in a *far field* at 915m with final concentration, C_f = 0,04mg/L.

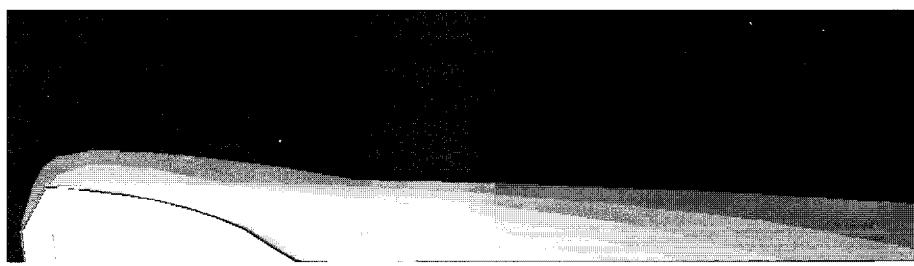

Fig. 1 Near field and limited value determined by legislation.

Fig. 2 Far filed showing the plume dispersion along the six transversal sections (915m far from ports).

REFERENCES

Cascardi, 2002. Monitoramento Ambiental do Emissário do Terminal Aquaviário de São Sebastião- Relatório Final. São Paulo.

Cetesb, 2001. Relatório técnico nº 010/01/EEAL, "Caracterização qualitativa dos despejos

líquidos do Terminal Marítimo Almirante Barroso - TEBAR Petróleo Brasileiro S.A. Petrobrás". São Paulo.

Fluent Inc, 2003. Version 6.1 (Tutorial).

Ortiz, J.P.; Bessa, K.L., 2004. Modelagem computacional de descarga de efluente em canal com análise dos campos próximo e distante. Anais do XXXI Congresso Latinoamericano de Hidráulica - IAHR. São Pedro, SP, Brazil.

Silva, L.S.; Miranda, L.B.; Castro, B.M., 2001. Relatório do projeto "Oceanográfico da Plataforma Interna de São Sebastião" (OPISS). Relatórios Técnicos do Instituto Oceanográfico. Universidade de São Paulo, pp 31.