An Overview of CFD Process for Application in Engineering Design and Analysis with A Case Study

A. Indra Siswantara
DaiCFD Research Group – Fluid Mech. Laboratory
Mechanical Engineering Department – University of Indonesia
Kampus Baru UI Depok 16425
e-mail: a_indra@eng.ui.ac.id

Abstract

This paper aims to provide an overview of CFD analysis process followed by case study of CFD application in hydraulic engineering. The case study considered is warm water dispersion in outlet channel of a coal-fired power plant. The numerical solution has been quantitatively compared with measurement data and showed favorable agreement. The prediction of temperature drop of 1^{0}C along the calculation domain provided an ample of safety margin to be used of reference in condenser design of the power plant. Lesson learned in CFD analysis applied in practical situation is also presented

Keywords: CFD, Numerical Analysis, Hydraulics

1. Introduction

Computational Fluid Dynamics (CFD) has become one of industrial standard tools for solving engineering problems. Lead time and cost reduction in engineering design and analysis benefit are among the factors that drive widely acceptance of this emerging technology in industry. However continuous research and developments are needed to enable CFD technology applications ease to use and provide fast and accurate solutions.

There exists a number of commercial CFD packages, general purpose CFD Code for solving fluid flow and heat transfer problems. These CFD packages come with advanced technology having their own features. For instance, seamless integration with advanced Computer Aided Design (CAD) and Computer Aided Engineering (CAE) code is one of advanced CFD code features offered by CFD vendor. Integrating analysis with the product design process which is one of the most talked about topics in the analysis community today. Users can take advantage of the existing data to conduct "what if?" analyses directly from within their chosen design platform. Software vendors, industrial users, and engineering managers all realize the vital importance of timely integration of analysis in the design process, but the visions, opinions and priorities placed on this can vary from one company to the next.

However the ability of the numerical method to provide a solution despite variabilities in the initial solution and control parameters or robustness is one of the most important factors to be considered when simulating fluid flow problem using CFD methodology. This incorporates issues of fault tolerance. Generally, robustness is achieved at the expense of accuracy (AIAA, 1998).

Most CFD Codes have sophisticated user interface, showing impressive colour graphics and animations. People can easily be led to an understanding that the computer solves all the problems with minimum knowledge of the user. Although present day computer programs can compute almost all problems, the accuracy of the results is still uncertain. An inexperienced user may produce convincing and impressive colour figures, but the accuracy of the results may still not be good enough to have a value in practical engineering (Olsen, N.R.B., 2004).

This paper aims to provide an overview of CFD analysis process and share CFD analysis experiences by presenting case study of CFD application in hydraulic engineering. The case study considered is warm water dispersion in the outlet channel of a coal-fired power plant.

2. CFD Analysis Process

The first stage of the analysis process is to formulate the flow problem. As with many analytical or computational problems it is worth to thinking about the physics of the problem. The CFD analyst should consider the flow problem and try to understand as much as possible about it. This can be achieved by seeking answers to questions: what is the objective of the analysis, what is the easiest way to obtain those objective, what geometry should be included, what are the freestream and/or operating conditions, what dimensionality of the spatial model is required? (1D, quasi-1D, 2D, axisymmetric, 3D), what should the flow domain look like, what temporal modeling is appropriate? (steady or unsteady), what is the nature of the viscous flow? (inviscid, laminar, turbulent), how should the gas be modeled, etc.

The second stage deals with modeling the geometry, flow domain and grid generation. This generally involves modeling the geometry with a CAD software package. Subsequently, decisions are made as to the extent of the finite flow domain in which the flow is to be simulated. In this stage the analyst has to calculate the grid or points or mesh that sub-divides the flow domain using mesh generator code.

The third stage is establishing the simulation strategy. The strategy for performing the simulation involves determining such things as the use of space-marching or time-marching, the choice of turbulence or chemistry model, and the choice of algorithms

The fourth stage is determining the flow specifications by establishing fluid parameters, boundary conditions and initial conditions. Since a finite flow domain is specified, physical conditions are required on the boundaries of the flow domain.

The fifth stage is performing the simulation. But, first the user must provide the information that will control the numerical solution. The simulation is performed with various possible with options for interactive or batch processing and distributed processing. As the simulation proceeds, the solution is monitored to determine if a "converged" solution has been obtained, which is iterative convergence.

The sixth stage is results analysis. In this stage the results is examined using post-processing code which also involves extracting the desired flow properties from the computed flowfield. The analyst checks that the solution is satisfactory. The computed flow properties are compared to results from analytic, computational, or experimental studies to establish the validity of the computed results. Analysis of the results also determines whether the process need to be repeated to examine sensitivities. The sensitivity of the computed results should be examined to understand the possible differences in the accuracy of results and / or performance of the computation with respect to such things as: dimensionality, flow conditions, initial conditions, marching strategy, algorithms, grid topology and density, turbulence model, chemistry model, flux model, artificial viscosity, boundary conditions and computer system

The seventh stage is documenting the analysis. Documenting the findings of an analysis involves describing each of these steps in the process.

3. CFD Case Study : Simulation of Warm Water Dispersion in Outlet Channel of Muara Karang Power Plant

3.1 Introduction

The rationale behind the study is that there is a need to confirm that the assumption of the temperature drop used in the previous simulation works is valid and there is common understanding that such study may be required in the future for envinromental study. The data for discharge characteristics is the same as the data used in the previous simulation work (A. Indra Siswantara et. al., 2004). The existing bathymetry data will be used as basis for extrapolation to obtain the sea bed topography of the discharge channel. A field sampling of the depth measurement has also been carried out to confirm the result of the extrapolation. The objectives of this study are to obtain numerical solutions of the 3D (three-dimensional) mathematical model of hydrodynamic and transport for the cooling water discharge in the discharge channel from PLTU out fall to the channel outlet at Pantai Mutiara and to predict the spreading pattern and temperature drop of the discharged cooling water from the PLTU discharge port to the Pantai Mutiara outlet using 3D thermal dispersion model with an assumed condenser temperature increase of 7.5^{o} C.

3.2 Existing Plant and Its Future Configuration

Muara Karang power plant is located in the Muara Karang area which resides within the larger Jakarta Bay area, located in the northern part of Jakarta. The area is located between 06^{o} 06’ 00” to 06^{o} 07’ 00” latitude and 106^{o} 47’ 00” to 106^{o} 48’ 00” longitude. The Muara Karang power plant complex is owned and operated by PT.Pembangkitan Jawa Bali (PJB) which is a subsidiary of PT. PLN (Persero). There are two power plant blocks in the Muara Karang bay area. The first block con-sists of 3×100 MW steam cycle units with oil-fired boilers (unit1, 2, and 3) and 2×200 gas-fired units (unit 4, 5). The second block is a Combined Cycle unit with a total capacity of 500 MW.

The steam cycle units with oil-fired boilers (units1,2,and3) will be refurbished and converted into combined cycle units with a total capacity of 740 MW class. This will consists of two (2) units of gas turbine and three (3) units of steam turbine. The capacity of the steam turbine with the new configuration will be 3×80 MW and the capacity of the gas turbine will be 2×250 MW.

3.3 Muara Karang Coastal Area

3.3.1 Seabed Bathymetry

The maximum depth of the sea bed is 8.1 meter at the northern area. The minimum depth is 1.0 meter at the western part of the shoreline. At the intake channel the water depth is ranging 3.3 m at the northern part of the channel to 2.1 m at southern part of the channel. The sea bed topology is in general sloped from the shore line with an inclination of about 0.0032^{o}.

3.3.2 Tidal and Tidal Current

The highest water spring (HWS) is 58 cm above the mean sea level (MSL). The lowest water spring (LWS) is 60 cm below the MSL. The MSL is located 223 cm above zero tide pole. The chart datum is located 60 cm below MSL. Based on the value of F of 3.4 and looking at the tidal elevation chart where only a single high and single low water occur each tidal day, the type of tide in Muara Karang coastal area can be characterized as diurnal type.

The tidal current observations during the period of September 2004 shows that the average of tidal current velocity is 0.061 m/sec with the predominant direction at 167^0. the average of non-tidal current velocity is 0.062 m/sec with the predominant direction at 175^0.

3.3.3 Meteorological Condition

Meteorological condition will mostly be based on the available Muara Karang AMDAL report. However, some additional data required for the computer model such as wet-bulb temperature, precipitation, and wind speed is obtained from Meteorological and Geophysical Agency. The ambient air temperature is relatively stable, with maximum temperature occurs in October and November, of 32.6^0 C and the minimum temperature in August at 21.4^0 C. The annual average temperature is 26.3^0 C. The highest relative humidity is 82% in January and the lowest is 72% in October. This data is taken from 10 (ten) years records of observation (1991-2000) as referenced in the AMDAL report. The rain fall level is relatively high, with the highest level of 451 mm in January, and lowest

25 mm in August. 10 (ten) years wind speed and direction data at nearest meteorological station in Tanjung Priok shows that wind speed is ranging from 2.4 knot to 13.3 knot, with predominant directions of wind are NorthEast and SouthWest depending on the monsoon season.

3.4. Modeling Approach

3.4.1 Computational Domain

The discharge channel has a rather complex geometry due to the shape of the channel which features constriction as well as expansion, and also a few islands situated inside the channel. This lands have to be modeled as wall boundary within the computational domain. The depth of the channel is obtained by extrapolating the existing bathymetry data on the adjacent coastal area. This extrapolation is required for building of the computational grid to be used for simulation.

The resulting extrapolated bathymetric data was also confirmed by a field sampling of sea depth measurement. The result of this field measurement will be used to ascertain whether the obtained data from the extrapolation of the existing bathymetry data is appropriate to be used for the computational domain preparation.

The extrapolated bathymetric data in form of the sounding coordinates and depths is converted into a GIS formatted file and used to generate the grid for the 3D model of the discharge channel. The situational layout of the power plant and the discharge channel to be simulated is shown in Figure 1 while the surface (topview) of computational domain and the grid arrangement can be seen in Figure 2. This grid has 19x22 cells with variable number of vertical layers depending on the depth of each cell which is determined from bathymetric data. Structural drawings of discharge channel were used to identify the location of the discharge port and other features of the channel so that the grid cell position can be matched with these features.

3.4.2 Field Measurement for simulation verification

A field sampling has been carried out during the period of the work to obtain a measurement of temperature and seabed bathymetry to be used for confirming the result of the computational domain to be used for the simulation and also the result of the simulation it self. A specified the sampling of temperature, sea depth measurement was taken at 3 (three) locations along the discharge channel.

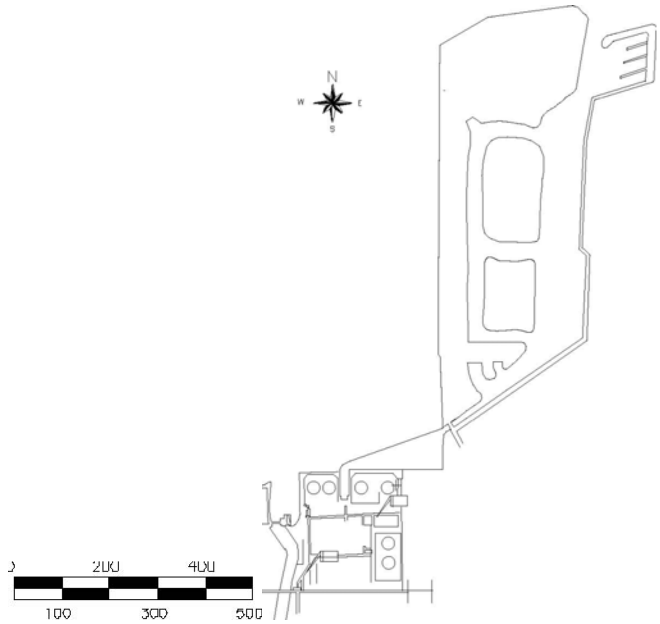

Figure.1. Situational layout of the power plant and the discharge channel to be simulated

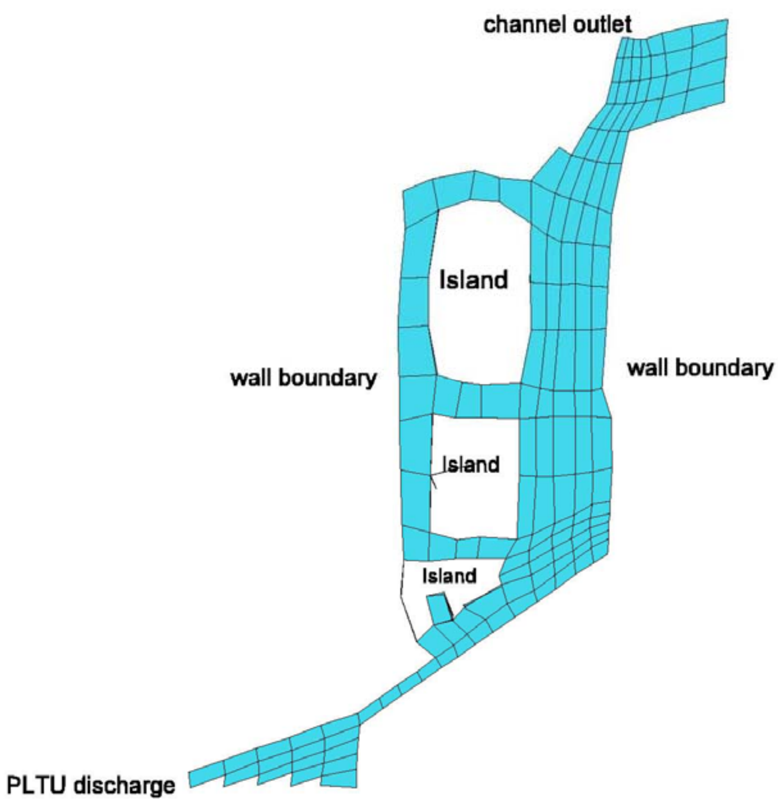

Figure2. Computational domain and grid arrangement

3.4.3 Simulation Scenario

Table 1 summarises the simulation scenario for the modeling of Muara Karang power plant discharge channel dispersion. The same flowrate condition were used in all scenarios. The sea water ambient temperature is taken as 30.5^{o}C, which is equal to the base (reference) temperature.

Table 1: Modelling scenarios with base temperature = 30.5^{o} C

Scenario ID	PLTU Discharge			Sea water
(Base Temp.	Flowrate	Δ T	Discharge	ambient
=30.5°C)	m^3/sec	°C	°C	°C
Scenario 1	38.2	7.5	38.0	30.5

3.4.4 Boundary Condition, Physical Parameters and Initial Condition

At the outlet of the discharge channel at the northside, open boundary condition is applied. The west and east side of the boundary wall boundary are applied which represent the wall of the channel. The transport equation at the open boundary are set with a Neumann condition, i.e.,no variation over space. The surface elevation is a function of time using the available water surface elevation data. The shore line is defined as wall boundary. The discharge from the power plant (PLTU) is defined with a constant volumetric flowrate (in m^3/sec) and are as listed in Table 1.

Weather data needs to be inputted which will then calculate several forcing terms at the air-water interface that will contribute to the result of the numerical computation. The forcing terms due to meteorological condition included in the calculation are the evaporation of the surface water and the wind shearstress. A time varying meteorogical data consisted of the following parameters is used during the period of simulation

The initial conditions define the state of the system at the beginning of simulation. As the intention is to calculate the scalar temperature out of the transport equations, the CFD code only needs the user to specify the initial temperature over the entire domain. Velocities are automatically computed/propagated from the surface elevation boundary condition. Other properties such as density are automatically calculated using the internal built-in function from the intial temperature efield. Inall of the scenarios to be run, a uniform temperature of 30.5^{0} C for the entire domain were specified. The simulation period is 13 days, starting from 3 Sep 04 to 13 Sep2004

3. 4. 5 Results and Discussion

In this chapter the simulation results will be presented together with the discussion of the results which will be emphasized on the respective temperature distribution pattern, and the magnitude of temperature of the discharged water as they travel from the discharge port to the channel outlet. The thermal dispersion pattern will be presented for both the ebb and flood condition. The ebb condition is represented by taking the thermal dispersion result on the 15 Sep 04 at 3:00 p.m. The flood condition is represented by taking the thermal dispersion result on the 15 Sep 04 at 09:00 a.m.

Model Validation

In order to assess the accuracy of the simulation model, a comparison of the simulation result with the data from the field measurement has been carried out. The data set of the simulation results to be compared with measurement was selected from a simulation time frame which will characteristically mimic the measurement time, in this case, at 12:00 in the afternoon. Table 2 shows the comparison of Surface temperature between simulation result and field measurement. It was shown that simulation results are in favorable agreement with the data from the measurement.

Table 2. Comparison between Simulation Results and Field Measurement

Position	Location Coordinate		Measurement	Simulation
No.	Latitude	Longitude	°C	°C
1	06 deg 04,925'	106 deg ,47.727'	37.0	37.4
2	06 deg 05,677'	106 deg ,47.697'	36.5	35.9
3	06 deg 06,154'	106 deg ,47.625'	35.7	35.2

Simulation Result during ebb condition

Figure 3. shows the thermal dipersion on the discharge channel during the ebb condition. During the ebb condition the discharged plume was directed to along the channel boundary to the outlet. The discharge temperature cooled from 38.0^{o} C to 37.5^{o} C as they travel within 300 meters from the discharge port. The discharge is then slowly cooled to 37.4^{0}C as it travels through the channel constrictions. The hot plume above 37^{o}C takes a straight line following the wall boundary pathline as it emerges from the constriction leaving the temperature in the area close to the island below 35^{0}C. At the main stem of the channel, the temperature is then cooled to 36.0^{0}C within 300 meter. The temperature at the outlet of the discharge channel is 35.04^{0}C.

Thermal Dispersion during flood condition

Figure 4 shows the temperature distribution in the discharge channel during the flood. Similar pattern is also takes place during the flood. The main difference from the ebb pattern is the area near the channel outlet. In this area the temperature the above 36.0^{0}C occupied a small area due to restriction of flow to the channel outlet caused by rising of sea level. This has caused the temperature in the channel in the vicinity of the channel outlet to decrease. Temperature at the outlet of the discharge channel is 34.95^0C.

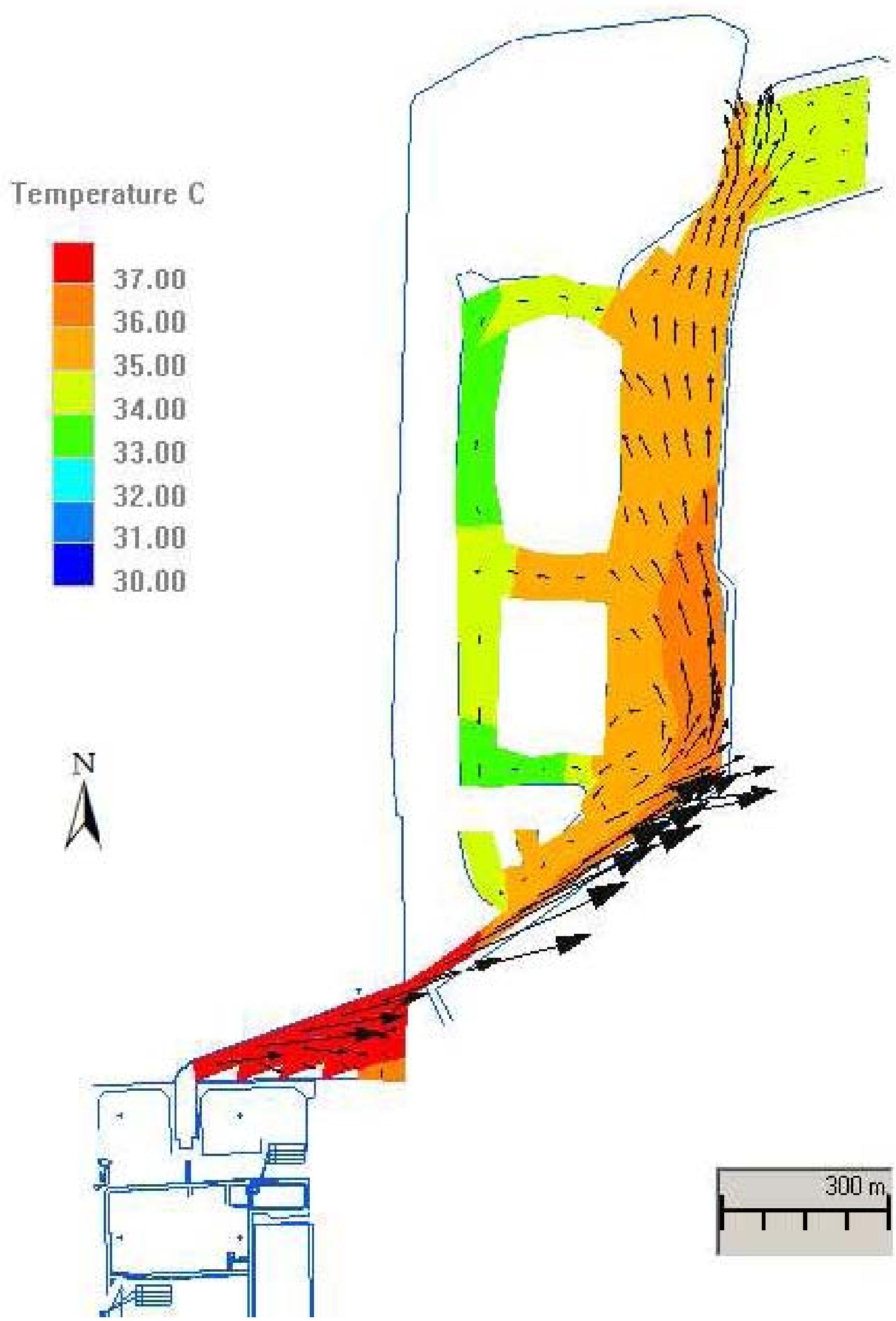

Fig. 3 Thermal distribution - ebb condition

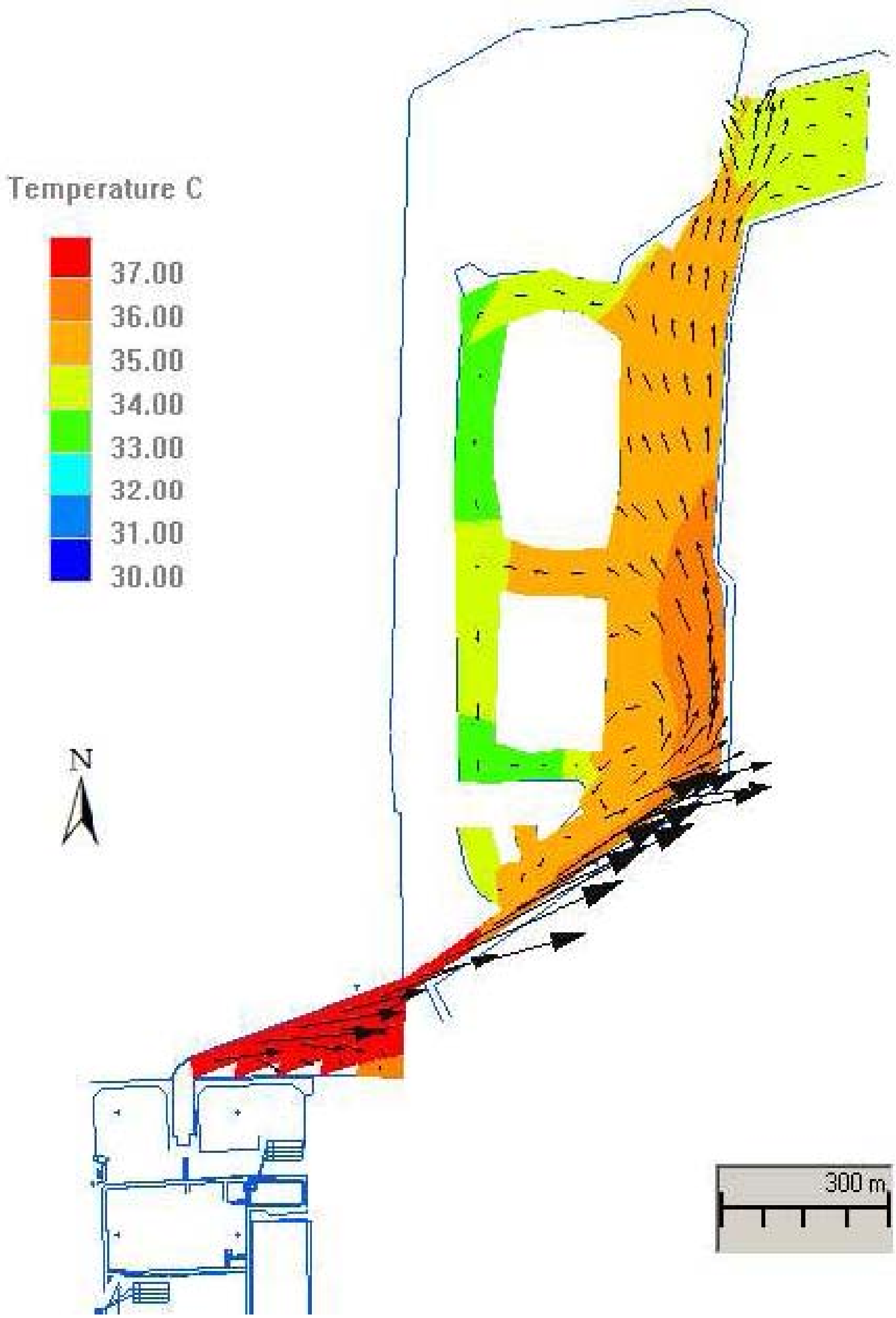

Fig. 4 Thermal distribution - flood condition

3. 4. 6 Conclusion

Numerical solution to the 3D hydrodynamic and transport model of the Muara Karang power plant discharge channel with thermal discharge sources from the PLTU was obtained.The numerical solution has been quantitatively compared with measurement data and showed favorable agreement. Thermal dispersion pattern of the cooling water discharge from Muara Karang PLTU has been simulated for a discharge temperature of 38^0C. The simulation for cooling water thermal dispersion on the discharge channel has been carried out for 13 days period. Results of the simulation shows that during the flood condition the temperature at the outlet of the channel is of 34.9^0C. During the ebb condition the temperature at the outlet of the channel is 35.04^0C. It is therefore concluded that the assumption of temperature drop of 1^0C used in the previous works is rather conservative and provide an ample of safety margin to be used of reference in condenser design.

4. Closing Remarks

1. An overview of CFD analysis process for application in practical situation with a case study in hydraulic engineering has been presented
2. It is important that the user of the computer programs has sufficient knowledge of both numerical methods and their limitation and also the physical process being modeled
3. CFD analysis process standard needs to be established. If it not possible to establish in the near future then it is necessary to have CFD analysis process guideline that is accepted by Indonesian CFD communities.

References

1. A. Indra Siswantara et. al., 2004, Simulation Study of Discharge Water Recirculation in Muara Karang Power Plant. Project Technical Report 2005. Jakarta
2. AIAA, 1998, *"Guide for the Verification and Validation of Computational Fluid Dynamics Simulations,"* AIAA G-077-1998.
3. J.Wu,E.M.Buchak,J.E.Edinger,V.S.Kolluru. 2001, Simulation of Cooling-Water Discharges from Power Plants. Journal of Environmental Management 61,77-92,2001.
4. Olsen, N.R.B., 2004, Hydroinformatics, Fluvial Hydraulics and Limnology, 4th Edition, Departement of Hydraulic and Environmental Engineering, The Norwegian University of Science and Technology.
5. PLN JE, AMDALReport. Muara Karang Gas Power Plant Project.
6. Shaw, C.T, 1992, Using computational Fluid Dynamics, 1st Ed., Prentice Hall International (UK).