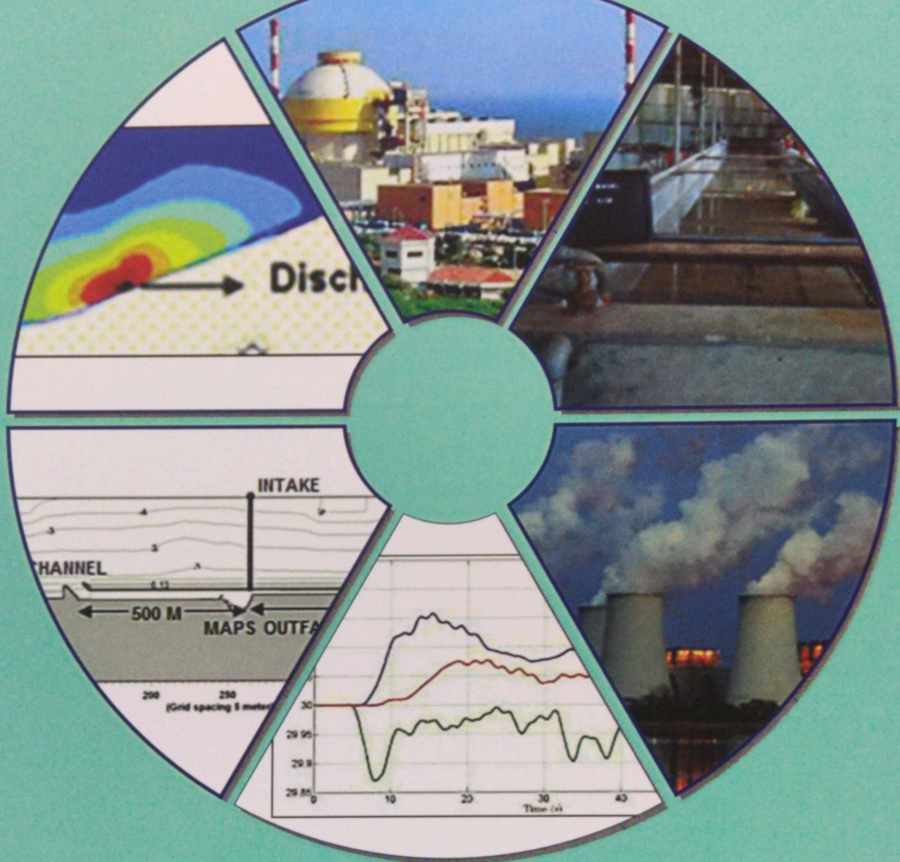
सत्यमेव जयते

भारत सरकार
जल शक्ति मंत्रालय
जल संसाधन, नदी विकास और गंगा संरक्षण विभाग

Government of India
Ministry of Jal Shakti
Department of Water Resources,
River Development & Ganga Rejuvenation



TECHNICAL MEMORANDUM
ON
GUIDELINES FOR LOCATION OF INTAKES AND OUTFALLS FOR NUCLEAR AND THERMAL POWER PLANTS

Dr. Jiweshwar Sinha, Scientist D
Shri S.N. Jha, Assistant Research Officer
Dr. Prabhat Chandra, Scientist E

100 years and beyond ...

1916-2016

Dr. (Mrs.) V.V. Bhosekar, Director

Central Water and Power Research Station
Khadakwasla, Pune 411024

JUNE 2020

Technical Memorandum

on

GUIDELINES FOR LOCATION OF INTAKES AND OUTFALLS FOR NUCLEAR AND THERMAL POWER PLANTS

Dr. Jiweshwar Sinha, Scientist D
Shri S.N. Jha, Assistant Research Officer
Dr. Prabhat Chandra, Scientist E

Dr. (Mrs.) V.V. Bhosekar, Director

Central Water and Power Research Station
Khadakwasla, Pune 411 024

JUNE 2020

FOREWORD

CWPRS provides services through research in water and power sectors. The services include field and desk studies, physical and mathematical model studies in these sectors. Commensurate with the past achievements, CWPRS has a great vision to be a centre of excellence in hydraulic engineering research and allied areas.

Hydraulic model studies gain a great importance in optimising the location of hydraulic structures. The rapid growth in the field of electricity generation through nuclear power projects has enabled us to use physical and mathematical model studies for finding the location of intake and outfall of such power projects based on thermal dispersion studies. For the location of intake-outfall, permissible limit for water temperature rise in a water body prescribed by MoEF is an essential requirement. In addition to this, siting of a power plant and various hydraulic, environmental and meteorological data required are also essential for carrying out dispersion studies.

Due to growing demand of electricity for the economic growth, India is on the threshold of a quantum expansion of its nuclear power programme. The nuclear and thermal power plants require large quantities of water for cooling purposes. The water taken from the source water body circulates through the plant and returns to the source through outfall with a higher temperature. Excess heat is an unavoidable by-product of electricity generation from fossil and nuclear fuels. The disposal of heated water from power plants and cooling systems into a natural water environment is a major environmental problem. The thermal efficiency of such power plants depends upon locations of their siting and discharge of warm water in water body. In absence of proper planning and design for the existing and upcoming coastal structures, the natural marine life at the site gets endangered and consequences of such development can be severe and long lasting. Therefore, it was felt appropriate to publish a Technical Memorandum on "GUIDELINES FOR LOCATION OF INTAKES AND OUTFALLS FOR NUCLEAR AND THERMAL POWER PLANTS" based on the studies carried out at CWPRS. The publication is expected to be useful for the modellers, designers and practicing engineers.

Pune
June 2020

Dr. (Mrs.) V V Bhosekar
Director

Executive Summary

The growing demand for energy to support and improve the quality of life throughout the world, coupled with the limitation of the world's resources of conventional fuels such as coal and oil, has required the development of alternative sources such as wind, thermal and nuclear power projects for electricity supply. Electricity is one of the key infrastructure elements for the economic growth of a country. The thermal efficiency of thermal and nuclear power plants in electricity generation is such that there is a need to dissipate a considerable amount of energy as waste heat. The discharge of this waste heat without disturbing the environment is a factor that must be taken into account in the siting of power plants particularly nuclear power plants, and this factor, is further growing in importance with the predicted rapid increase in nuclear power production throughout the world.

Many power plants require large quantities of water for cooling purposes. The water taken from the source water body (e.g., lakes, estuaries, bays, open sea and rivers) circulates through the plant and returns to the source through outfall with a higher temperature. Excess heat is an unavoidable by-product of electricity generation from fossil and nuclear fuels. The disposal of heated water from power plants and cooling systems into a natural water environment is a major environmental problem. Environment plays a vital role in overall development of a country. For optimal performance of the power plant, the intake inlet and discharge outlet should be meticulously placed so that the heated water will not re-circulate back into the power plant. The cooling water of the thermal power plant is considered one of the most polluting sources of water bodies which requires large amount of refrigeration water. Thermal pollution affects both the physical and chemical characteristics of the flowing water. Also, thermal pollution effect on morphology of the water bodies due to the growth of the water plants depending on the water temperature increase and sedimentation process.

The discharging of the cooling system of the thermal or nuclear power stations into water bodies may be surface discharge, submerged point discharge and submerged multi-port diffusers. A once-through cooling power plant draws huge amount of water each day from nearby lakes, rivers or oceans. The withdrawn water is boiled and turned into steam, which spins the turbines of power plant in order to generate electricity. In a power plant, intake and discharge systems are one of the most important components, as they deliver cold water to the plant and discharge the hot water back to the source. Their locations should be analyzed meticulously to reduce the rise in water temperature at the intake to permissible values. Thermal recirculation studies are frequently used to demonstrate that the thermal limits are met. The heated water released from the plant outlet transports and disperses in the source water body by a combination of all available hydrodynamic forces.

Various methods exist to determine the size of a temperature field from a thermal discharge. Because of the difficulties in simulating actual turbulent jets into real environment, all of these methods are approximate. They, however, can be used to predict temperatures with reasonable accuracy. Each method has its own advantages and disadvantages.

Physical modelling is a technique where a scale model of the receiving water and discharge system is constructed to carry out measurements that can be used to estimate the response in the actual discharge. Dynamic, kinematic and geometric similarity laws are used to determine size, shape and operating conditions of the model. Dynamic similarity requires that various dimensionless parameters such as Reynolds number, Froude number and Nusselt number be the same for both the physical model and full sized discharge and environment. Further, scale effects arise in physical modelling due to force ratios which are not identical between a model and its prototype and result in deviations between the up-scaled model and prototype observations. However, by using Mathematical modelling technique, the scale effects can be avoided, compensated or corrected. The advantage of physical models is the ability to include time dependency and the complete geometry of the environment and discharge. The disadvantages are the high cost and the inability to look at variations with ease.

Mathematical modelling of thermal discharges aids in the prediction of possible effects which may result from changes in physical properties of water due to the extent of the influence of thermal discharges. Modelling provides a technical basis for developing a discharge structure design which disperses the heat effluent in a manner which will satisfy temperature standards and also estimates the possible recirculation of heated water back into the intake structure. To study the circulation features in the study area at base line condition, Hydrodynamic (HD) Module is used. Thermal and Salinity Dispersion Study are carried out using Advection-Dispersion (AD) module. The various parameters need to be considered in Advection-Dispersion (AD) module are circulation features from the Hydrodynamic (HD) Module (Water level, Currents) and the characteristics of the discharge outfall (Temperature, Salinity). The governing equations for warm water are generally solved using different numerical methods such as finite difference method (FDM), finite element method (FEM), spectral method and finite volume method (FVM) as in the laterally integrated hydrodynamics and mass transport model. The independent variables are longitudinal distance, flow depth, and time. The water surface elevation and horizontal momentum are computed simultaneously in the model based on a numerical scheme, which allows the use of a reasonable time scale for field application over entire stratification cycles. Mathematical models are generally developed for one-dimensional, two-dimensional and three-dimensional free surface flows. There are a number of commercial and scientific software available for this like MIKE, Flow3D, Telemac, OpenFOAM, SU2, Delft3D, EFDC etc. The advantages of mathematical methods are the ability to represent actual geometry and time dependency without having to go to the laboratory and set up a physical model. Disadvantages include the time required to set up and run a problem having a complicated boundary and the inability to accurately represent turbulence numerically. Herein we have taken up MIKE software for discussion on this. Numerical model software, MIKE21AD simulates dispersion of warm water in coastal environment. The modelling software is developed by Danish Hydraulics Institute (DHI), Water and Environment, Denmark and is one of the widely used commercial software.

Chapter 1 describes briefly both the worldwide and the Indian scenarios of electric power generation, uses and their future needs. The requirement of electricity for sustainable economic development is highlighted. Different Non-conventional and conventional sources of electricity generation have also been discussed.

Chapter 2 deals with the theories adopted in dispersion of warm water from the thermal and nuclear power plants. Design of intakes and outfalls have been analysed and different equations used for thermal discharges are enumerated in detail.

Chapter 3 consists of requirement of hydraulic, environmental and meteorological data for fixing the location of intakes and outfalls of thermal or nuclear power plants. The importance of accuracy of these data has also been discussed.

Chapter 4 describes various modelling techniques generally adopted for finding the location of intakes-outfalls structures in a water body. Among the different modelling techniques such as physical model, empirical model, integral model, mathematical model etc. mathematical model techniques have been discussed in detail. In physical modelling, various techniques are adopted to control the dynamics of flow and to ensure turbulent Reynold's number. In mathematical modelling, MIKE 21 and MIKE 3 software developed by DHI, Denmark are generally used at CWPRS for finding the location of intake and outfall of a power plant based on thermal dispersion studies. The governing equations and the numerical techniques used for dealing with these equations in MIKE software are also detailed.

In Chapter 5, various case studies of physical model studies and 2-D & 3-D mathematical model studies undertaken for both thermal and nuclear power plants have been described. These studies are successfully completed at CWPRS for locating intakes and outfalls based on warm water dispersion for various thermal and nuclear power projects have been enumerated. Major studies carried out are NCTPS, ETPS, Kalpakkam, Kudankulam, Jaitapur, Kovvada, Tarapur etc.

CONTENTS

Chapter 1

INTRODUCTION

Energy is an immensely important factor for sustainable economic growth and improved human life. Rapid economic development and worldwide population growth demand an increasing number of power plants. Energy is produced from various sources and the increasing trend in the energy production worldwide is shown in Figure 1.

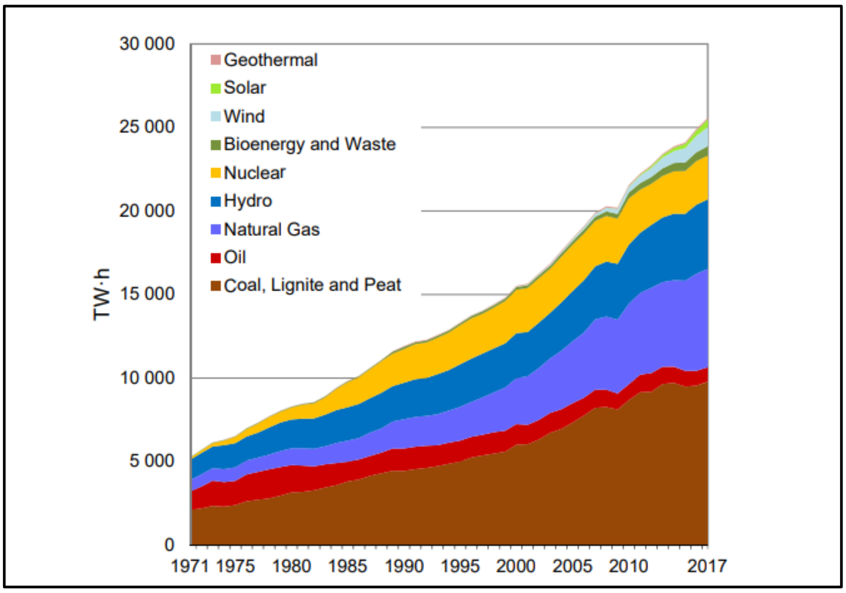


Fig. 1 Breakup of world total electricity production by energy source during the period 1971–2017 [20]

Out of these sources, thermal power generation is the traditional method for energy generation in which turbine is operated by steam, by burning coal, gas or by heating water. Nuclear Power Plants make use of the heat produced from nuclear reactor to run steam turbine. The burning of fossil fuels affects the environment by producing several pollutants like CO_2, NO_x, SO_x, fly ash etc., whereas Nuclear power proves to be a clean, reliable and affordable energy source which lessens the negative impacts of climate change comparatively. It contributes to a major part of the total world energy and its use is growing extraordinarily with time. At the end of 2017, there were 448 operational nuclear power plants in the world, having total net installed power capacity of 392 GWe. Only the operational nuclear reactors contributed to about 1% increase in electricity generation in 2017, thereby increasing the number to 2503 TW·h [52]. Nuclear power alone accounted for about 10% of the total electricity produced in 2017. World's nuclear electricity production in

2017 is shown in Figure 2. Power generation during 2017-18 using different sources is shown in Figure 3.

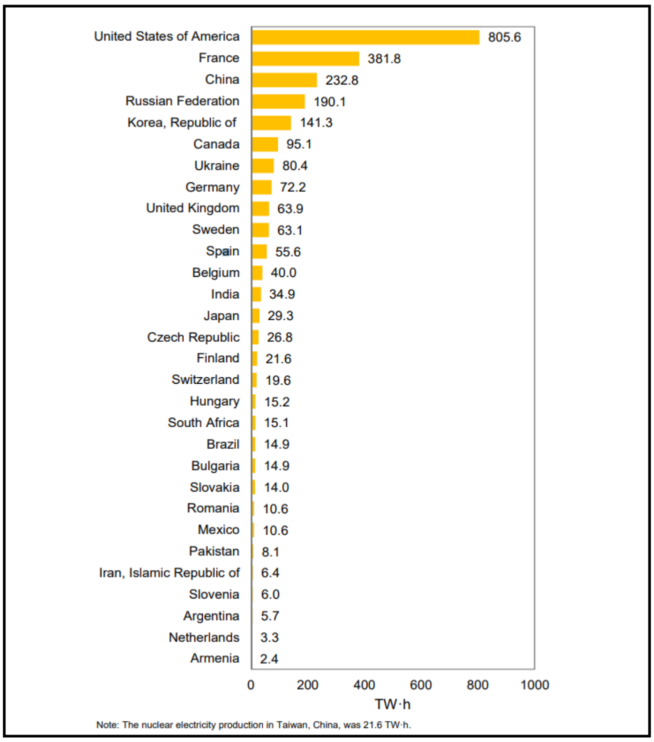


Fig. 2 World nuclear electricity production in 2017 [20]

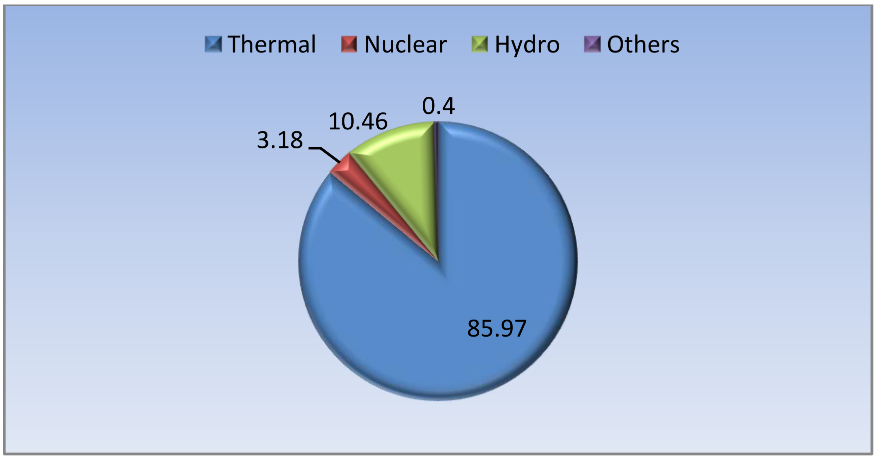


Fig. 3 Power generation during 2017-18 (%) [11]

For Indian scenario, the total installed capacity using thermal power plants is of the order of 226.27 GW and Electricity Generation in March 2019 is 94.83 BU. During that period electricity generated by thermal sources was 63.7% of the total Electricity Generation. Similarly, the installed capacity by nuclear power generation is of the order of 6780 MWe and the increase in Electricity Generation is 2.69% during the year 2018 [11]. The power generated is 1.9% of the total Electricity Generation. The electricity generation increased from 771.51 BU in 2009-10 to 1212.134 BU in 2017-18 and is shown in Figure 4. Although India is the fifth-largest producer of electricity, the per capita consumption of electricity, which has a direct correlation with the Human Development Index (HDI), is very low at about 700 kWh per annum. As electricity is a key driver for economic growth, it is necessary that there is a massive augmentation in electricity capacity. The Integrated Energy Policy of the country projects the need for an installed capacity of about 778 GW by the year 2032 for a growth rate of 8%, of which nuclear power is envisaged to be about 63 GW by 2032 [48]. However, in view of India's energy resource profile, it is inevitable that the long-term electricity needs have to come from nuclear energy.

At present India is on the threshold of a quantum expansion of its nuclear power programme. There are plans to significantly increase the installed capacity in nuclear power by setting-up large light water cooled reactors of overseas design, apart from the indigenously designed PHWRs and Fast Breeder Reactors. The nuclear power programme in India is being pursued with full regard for nuclear and radiation safety, which encompasses safety of public and the environment around the plants. The primary responsibility of ensuring safety of NPPs rests with the organisation responsible for the design, construction and operation of NPPs. In India, these activities are carried out by the Nuclear Power Corporation of India Ltd (NPCIL) and the Bharatiya Nabhikiya Vidyut Nigam (BHAVINI). The task of laying down necessary safety requirements and their enforcement are entrusted to the Atomic Energy Regulatory Board (AERB).

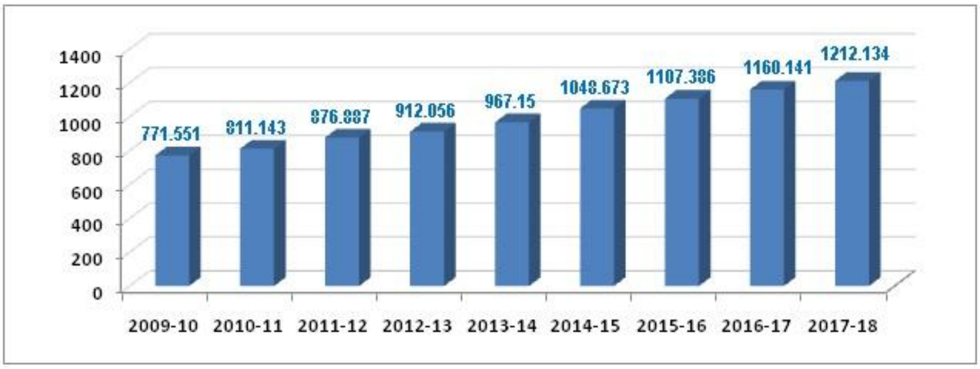


Fig. 4 Electricity Generation (Billion Units) 2009-2018 [11]

However, the undesirable temperature of effluent water from thermal or nuclear power plant is a common concern to people, industrialists, environmentalists, regulatory agencies etc. inhabited in the vicinity of the plant. Coastal development and hence the inevitable associated heat build up in surrounding water body can potentially disrupt vital ecological pathways and cause damage to nourishing fields of commercial shellfish and fish species. In absence of proper planning and design for the existing and upcoming coastal structures, the natural marine life at the site gets endangered and consequences of such development can be severe and long lasting.

Despite the need, there is relatively less guidance available regarding thermal dispersion problems. Before the implementation of Environment protection Act, 1986, Environmental Clearance for projects was not mandatory. According to Environment protection Act, 1986, the temperature raise caused by the condenser cooling water discharge from power plant to the receiving water body should not be more than 5 ^{0}C higher than the intake water temperature [34, 37]. Consequently, it is amended vide 1991 act of Ministry of Environment and Forest (MoEF) that the effluent discharge should not be more than 7 ^{0}C above the ambient temperature. Secondly, if the warm water discharged through the outfall of power plant enters the intake, it leads to loss of efficiency of the power plant.

The generation of waste heat is an unavoidable by-product of the commercial generation of electricity from fossil and nuclear fuels. Historically, these waste heat streams have been handled by transferring the heat to a cooling water stream, typically withdrawn from a nearby water body, and then returning the heated water to the environment. The discharge of heated water in a water body causes the rise in temperature of water surrounding outfalls which affect the aquatic organisms. Raising or lowering the water temperature may lead to the ecosystem disorders with the impacts on water quality, impacts on micro-organism and impacts on fishes.

In order to avoid the water temperature rising too fast, the load of the exhaust heat borne by water should be limited within a certain range. The location of intakes and outfalls should be designed rationally. The reasonable location of intake and outfall is not only conducive to the diffusion of the thermal discharge but also not cause the secondary heat return. The arrangement of intakes and outfalls should take the effect of the following aspects into consideration.

1. The higher the temperature of intakes, system's cooling performance is lower, so one should avoid the thermal discharge affecting the intakes' temperature.

2. One should try to narrow the distance between intakes and outfalls to reduce the energy consumption.

3. The standard to judge if the temperature of surface water rises or drops should be outlined strictly.

The dispersion processes and the related morphological evolution due to thermal discharge in water body can be described mathematically by solving partial differential equations, formulated in variables such as velocity, pressure and surface elevation. These equations are continuous in space and time. But, in practice, these differential equations cannot be solved analytically. In solution to these, numerical models offer an effective alternative. They can transform any general equation into numerical algorithms. These equations are coded into algorithms and used as an input for computer programs. The computer programs give numerical solution which is in quite concurrence/accordance/agreement with the exact continuous solution.

There are many advantages of numerical or mathematical models. These are fast, reliable, less costly and powerful tools. The models help analyzing the given problem. They are also used to predict the effects of different solutions offered after carrying out various trials to arrive at the best optimal solution to the problem. The stratification effects over the water vertical due to the difference in densities is also well considered by these models. However, the numerical models, which are the approximations of reality are just a helping tool for an engineer/scientist. They cannot completely replace the knowledge and the analytical abilities of coastal experts.

The outputs of the mathematical model used for the problem under discussion help us to predict the spatial and the temporal domain in which the increased temperature will impact the aquatic life. It also suggests the alternate locations that can be considered for better dispersion of the heated discharge. Further, intakes of the power plant are strategically located based on the results of mathematical model studies to avoid re-circulation of warm water in the plant.

In this technical memorandum on 'GUIDELINES FOR LOCATION OF INTAKES AND OUTFALLS FOR NUCLEAR AND THERMAL POWER PLANTS', an attempt is made to provide the guidelines for locating the intake-outfall structures of thermal and nuclear power plants based on thermal dispersion problems of warm water discharged from these power plants using mathematical modelling. In the first chapter of this memorandum, introduction about the topic covering thermal and nuclear power plants is mentioned. In the second chapter, theory and analysis involved in the process is discussed. Need of relevant data for the locations of intake-outfall of the power plant is enumerated in chapter three. Different mathematical modelling techniques required for locating the suitability of intake-outfall of the power plant are detailed in chapter four. In the chapter five, some case studies carried out at CWPRS are discussed. Finally, the conclusion part of the Technical memorandum is added at the end.

Chapter 2

THEORY AND ANALYSIS

2.1 Introduction

Thermal power is the largest source of power in India. About 71% of electricity consumed in India is generated by thermal power plants. Nuclear power is the fifth-largest source of electricity in India after coal, gas, hydroelectricity and wind power. As of March 2018, India has 22 nuclear reactors in operation in 7 nuclear power plants, having a total installed capacity of 6,780 MWe. Nuclear power produced a total of 35 TWh and supplied 3.22% of Indian electricity in 2017 [38]. Seven more reactors are under construction as on today with a combined generation capacity of 4,300 MWe. Thermal and nuclear Power Projects are preferred over Hydroelectric power projects now-a-days because of their small gestation period and relatively less rehabilitation and resettlement issues.

Table 1. Nuclear power generation

Year	Generation (TWh)
2006	17.7
2007	17.7
2008	15.0
2009	16.8
2010	23.0
2011	32.3
2012	33.1
2013	33.1
2014	34.5
2015	38.4
2016	38.0

2.2 Basic working principle of a typical thermal power station

Generally bituminous coal is used as the fuel in coal based thermal power plants. It is transported from mines to the plant. It is pulverized and mixed with preheated air in the boiler which results into formation of heat emitting fireball. The heat, thus produced is used for converting water into steam at high temperature and pressure. This saturated steam is further heated to 540^{0}C in the super heater. Steam turbines are run using this superheated steam. The high quantum of energy present in the saturated superheated air converts into mechanical energy in the turbine because of rotation of turbine blades. Turbines are coupled with alternator, which in turn generate electricity [13]. Electricity is transmitted at the desired locations. Meanwhile, there is a drop of temperature and pressure when the steam passes through turbines and the temperature of the water, thus produced is further brought down in the condenser. Condensing is a very essential process, as even the resulting water is of quite high temperature which cannot be released to the water source as it is. A simplified layout of a typical thermal power station is depicted in Figure 5.

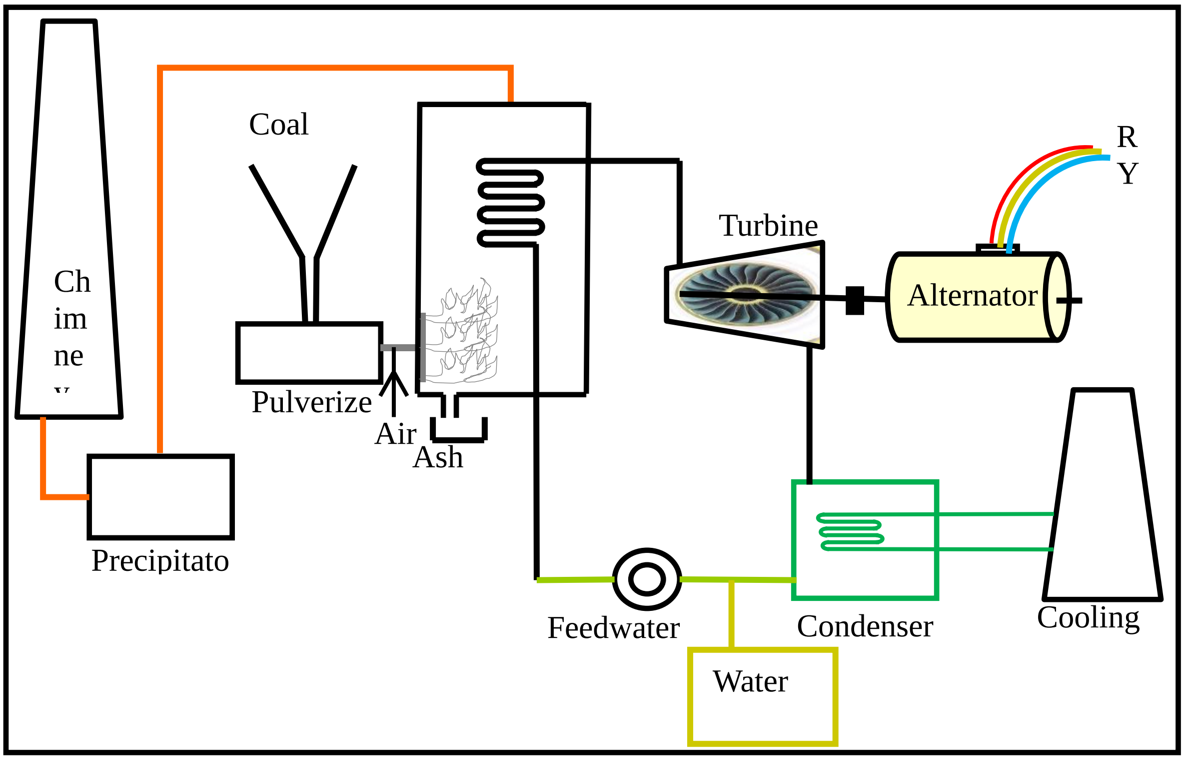


Fig. 5 Simplified Layout of a typical thermal power station [13]

2.3 Basic working principle of a typical nuclear power station

Steam is generated in nuclear power plants by heating the water through the energy derived from the nuclear fission reaction. A nuclear power plant consists of following components [14] as indicated in Figure 6.

Fuel

Uranium is the basic fuel. Usually pellets of uranium oxide are arranged in tubes to form fuel rods. The rods are arranged into fuel assemblies in the reactor core.

Moderator

Material in the core which slows down the neutrons released from fission so that they cause more fission. It is usually water, but may be heavy water or graphite.

Control rods

These are made with neutron-absorbing material such as cadmium, hafnium or boron, and are inserted or withdrawn from the core to control the rate of reaction, or to halt it. Secondary control systems involve other neutron absorbers, usually boron in the coolant – its concentration can be adjusted over time as the fuel burns up.

Pressure vessel or pressure tubes

Usually a robust steel vessel containing the reactor core and moderator/coolant, but it may be a series of tubes holding the fuel and conveying the coolant through the surrounding moderator.

Steam generator

Part of the cooling system of pressurised water reactors (PWR & PHWR) where the high-pressure primary coolant bringing heat from the reactor is used to make steam for the turbine, in a secondary circuit. Reactors have up to six loops, each with a steam generator.

Containment

The structure around the reactor and associated steam generators which is designed to protect it from outside intrusion and to protect those outside from the effects of radiation in case of any serious malfunction inside. It is typically a metre-thick concrete and steel structure.

A simplified sketch of a typical nuclear power plant is depicted in Figure 6. Nuclear power plants use the energy in the superheated steam to run the turbine blades with only difference that the heat required for steam production is obtained from exothermic fission reaction of radioactive materials [32]. Generally, the radioactive material used is Uranium-235 which is a highly unstable atom. It is made to strike with a thermal neutron. After fission, more unstable atom, Uranium-236 is produced. It splits into very stable Krypton-92 and Barium-141 as a by-product of this reaction, 3 more neutrons are produced and heat energy is released. These free neutrons again react with other uranium atoms and thus, the chain reaction starts, releasing of a great amount of thermal energy [52]. In order to bring down the high energy neutrons to the optimum speed, moderator (heavy water) is used in the high pressure vessel.

This water partially absorbs the heat energy. Water from a water source is introduced into the heat exchanger. It absorbs the heat of the moderator and converts it into highly pressurized steam. This steam is used to run turbine blades as well as to run the armature of generator thereby generating electricity, which is transported through transmission lines to area of requirement. Hot steam from the turbines condenses through the condenser and the warm water thus formed, is cooled through the cooling tower as it cannot be released to the water source as it is because of its raised temperature.

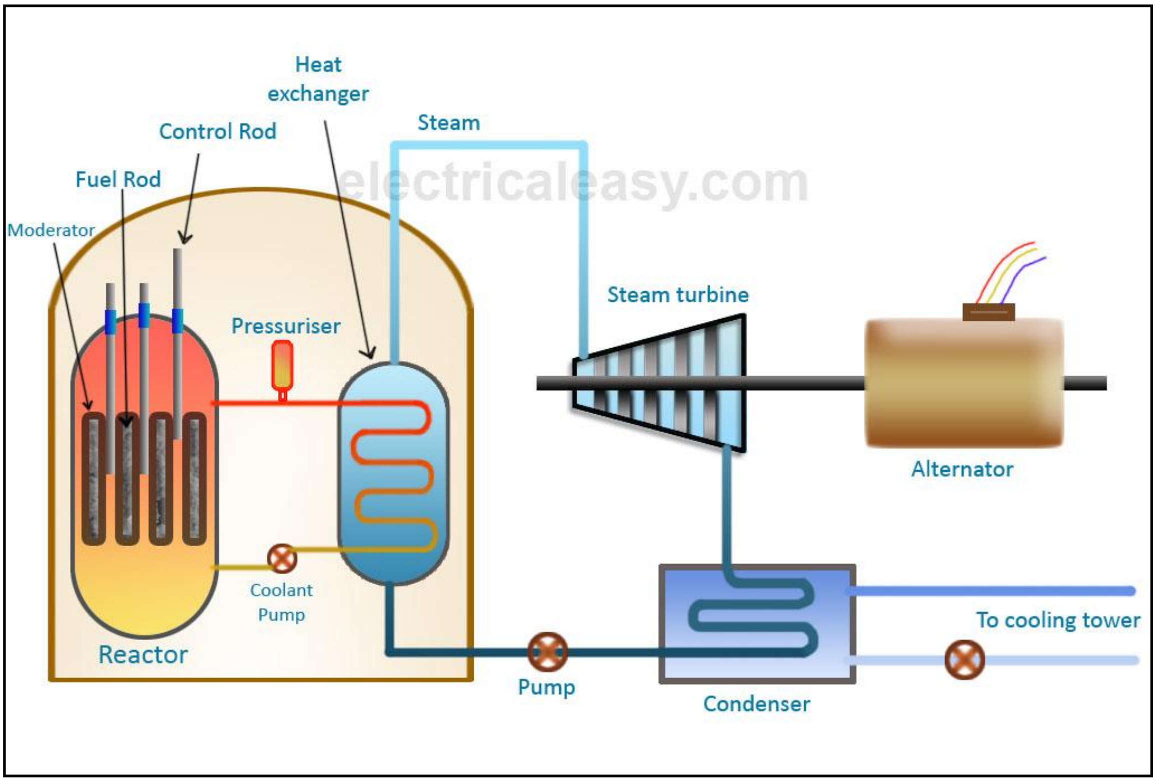


Fig. 6 Sketch of a typical nuclear power station [14]

The first nuclear power plant in the country, comprising two nuclear reactor units, was set up at Tarapur, Maharashtra on turnkey basis by GE, USA. The construction of these units began in 1964 and they became operational in October, 1969 [19]. Subsequently, work on the Pressurised Heavy Water Reactors (PHWRs) of the first stage began with construction of RAPS-1&2 at Rawatbhata, Rajasthan. Subsequently, MAPS units-1&2, NAPS-1&2, KAPS-1&2, Kaiga-1&2 and RAPS-3&4 were set up with indigenous efforts. In parallel to the indigenous three-stage programme, based on international technical cooperation faster nuclear power capacity Two Light Water Reactors (LWRs) of 1000 MW each are constructed at Kudankulam in technical cooperation with the Russian Federation. Further expansion with four more units of nuclear power generations are underway at Kudankulam.

There are presently 19 nuclear power reactors in operation with a capacity of 4560 MW. The 20th reactor, Kaiga-4 (220-MW) has the nuclear power installed capacity to 4780 MW. In addition, three nuclear power reactors with a capacity of 2500 MW are at an advanced stage of construction and four reactors each of 700 MW, two each at Kakrapar in Gujrat and Rawatbhata in Rajasthan, respectively, have also been launched. With the completion of the reactors under construction, the nuclear power capacity in the country will reach 7280 MW and 10080 MW [21]. India today is recognized as a country with advanced nuclear technologies.

2.4 Heat rejection to surface water

The discharge of thermal effluent from the power plants into water bodies would result in harmful impacts on the ecological life as discussed earlier. The higher forms of aquatic life require oxygen for survival. The high temperature decreases the concentration of oxygen in water. So, it is important to dilute the thermal concentration into water bodies and confine it into small areas to maintain the allowable limits of oxygen needed for the aquatic life. When waste heat carried by condenser cooling water is discharged into natural water bodies such as rivers, lakes, or coastal waters, the transfer to the atmosphere occurs over relatively large areas by evaporation, radiation, convection and conduction.

Heat dissipation from the receiving water surface is naturally return the water to its natural temperature state within a certain distance from the point of heat discharge. This distance depends on a number of processes, e.g. the amount of mixing or dilution between the heated condenser water discharge and the receiving water and the transfer of heat from a water surface to the atmosphere through the combined mechanisms of evaporation, radiation, convection, and conduction. The percentage of the total heat dissipation by evaporation increases as the temperature of the water surface increase above the air-water interface equilibrium temperature. The smaller the heat loss by evaporation, the lower the consumptive use of the water [24].

The proper design of discharge structures for once-through systems is an important factor in determining the magnitude, extent, and distribution of thermal effects in the receiving water bodies. There is a high degree of flexibility in tailoring the temperature distribution in the receiving water to minimize the biological impact. At opposite ends of the design capability are complete stratification or complete mixing of the heated effluent.

In case of stratification, mixing is avoided and the heated water is floated onto the receiving water in a relatively thin surface layer. Heat transfer to the atmosphere is at a maximum rate, and there are no temperature change at or near the bottom of the receiving water, as long as there is no significant turbulence. Further, the strong temperature gradients at the plume boundary can be detrimental to some aquatic organisms. For rapid mixing of the warm water into the whole water body, the condenser cooling water is conducted through a diffuser pipe or tunnel and discharged through nozzles or ports near the bottom of the waterway [24]. Entrainment of surrounding water into the high velocity jets produces rapid dilution, with minimum temperature gradients. Being the simplest and least expensive cooling technique, once-through cooling remains the first choice, whenever applicable. The environmental advantages of this method are the low consumption use of water, the ability to tailor the temperature distribution field in the receiving water to meet biological and temperature objectives and heat dissipation to the atmosphere. Figure 7 shows schematically a cooling system where water is simply taken from a surface water body, passed through the condenser tubes and then returned. The water bodies generally used as a source of cooling water to the power plants are enumerated below.

2.4.1. Rivers, streams and canals

When a river or stream is used as a source of cooling water, the water is discharged downstream of the intake. Provided the flow-rate is sufficient to satisfy environmental requirements, this method consists in using the water surface downstream as a natural heat-exchange surface to the atmosphere.

The goal of river mixing studies is generally to estimate the longitudinal and transverse dispersion coefficient or to investigate the effect of different parameters such as river discharge, bed roughness, ice cover, etc. on the mixing potential [49]. The longitudinal and transverse dispersion coefficients are both based on the Fick's law and describe the elongation of the tracer cloud in stream wise or transverse direction, respectively [16].

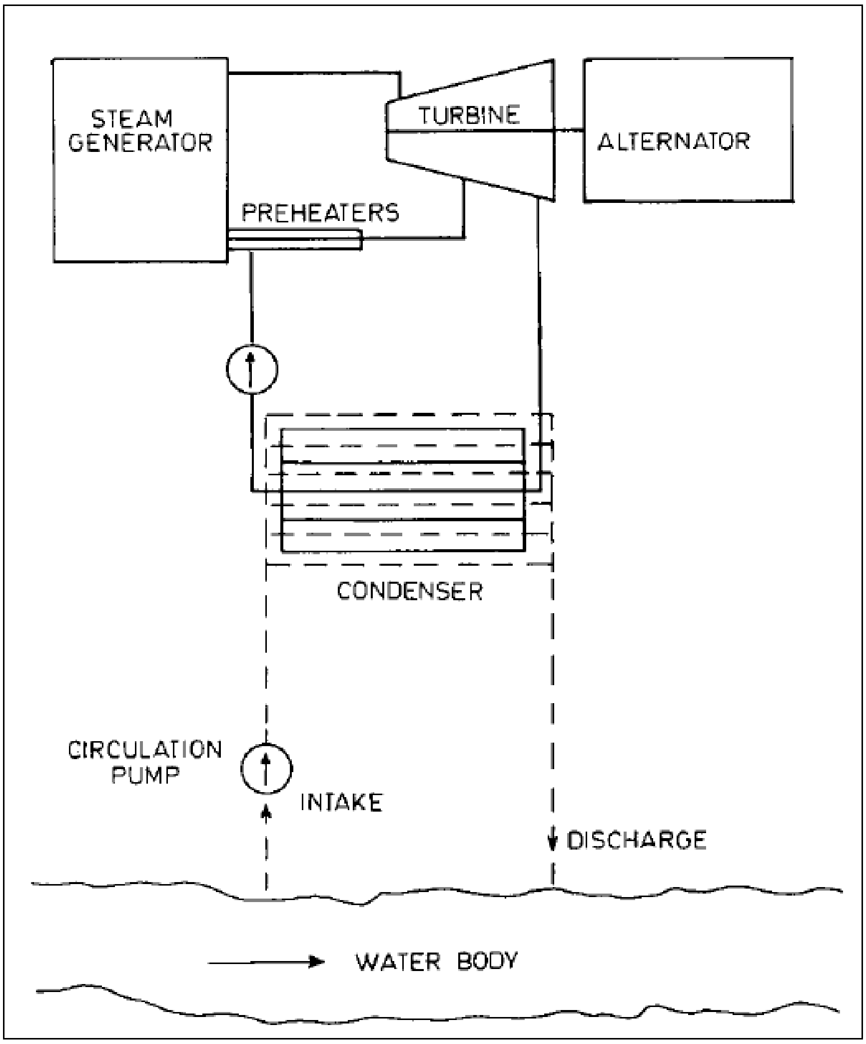


Fig. 7 Outline of power plant with direct intake and discharge in a water body [19]

2.4.2. Ponds, reservoirs and lakes

Cooling ponds, reservoirs and lakes are water bodies which in general are used exclusively for power plant cooling. Two types of cooling pond operation are possible: recirculation or once-through. In the recirculation type the condenser cooling water is discharged at one end of the pond and pumped at the other end, thereby forming a closed system with the power plant. The water surface area determines the pond effectiveness to deliver cold cooling water to the condensers. The area of a closed cooling pond required to reach a given cold water temperature can be estimated in advance.

2.4.3. Estuaries and coastal sites

Sea water offers good potential as coolant for electrical generating plants. Thermal power plants located on sea coasts generally use sea water as the source of raw water. Sea water is conveyed to the plant through the sea water pipelines from intake structures. The sea water is used as cooling water. The cooling water systems for power plants could be of once through type or re-circulating type. The decision for the type of cooling water system is done considering environmental stipulations, site condition, availability and ease of drawl of water, local regulations and evaluation of techno-economic aspects. Re-circulating type sea water cooling system consists of an intake, pump house and pipe lines to convey sea water to the plant [41]. In addition to these structures in a once through cooling system there will be an outfall and pipelines / channels to convey reject water to the outfall. When due care is taken to avoid local effects in the discharge area, the ocean offers the possibility of large-scale heat rejection with a small environmental impact. Once-through cooling with sea water is already extensively used, especially in countries with a small river run-off and short access to coastal sites, such as Japan and the United Kingdom.

There are few problems associated with cooling water discharges to the sea along an open coastline. The mixing of effluents depends upon tidal movements, shore currents and wind driven circulation. The local hydrographic conditions need to be studied and precautions taken to avoid recirculation of the discharged water. Specific features of coastal mixing studies include the mixing sources, such as tidal currents, waves and wind, which affect the dynamics and the mixing pattern of the plume. The wind can influence coastal mixing through wind generated waves or wind driven currents. Density stratification of the ambient receiving water can also be an important consideration. Turbulence is the main source of mixing and can be produced by fluid shear, friction or buoyancy.

In view of the above, seawater Intake and Outfall system design involve:

a) Sea bed profile is a critical input that decides the type of Intake
b) Oceanographic Data Collection
c) Data regarding tide levels, tidal currents, wave conditions and storm surges are critical basis for designs for the intakes and outfalls
d) Geotechnical Investigation & Surveys

e) Both off shore and on shore geotechnical investigations are carried out to get soil data which is required to decide on the location of various structures and to decide the type of foundation.

f) Environmental Impact Assessment

The impact of the intake /outfall structure on marine life during construction stage and during the life time of the project needs to be studied.

2.5 Thermal discharge

Thermal discharge can be classified according to the location of the discharge pipe, the discharge pipe dimensions and cross section, the number of diffusers and so on. The main two types of thermal discharge into shallow and still receiving water based on locations are submerged and surface discharge [29].

2.5.1 Submerged Discharge

The structure design of the discharge pipe is varied from one site to another, because there are different types of structure. The discharge pipe generally locates below the free surface of the water body with a certain depth. This type of thermal discharge is complex because the discharge pipe is submerged below the free surface of the receiving water and the thermal plume cannot be observed. In thermal submerged discharge the plume is generally influenced by the free surface and the bed. It may get deflected towards the free surface with increasing distance downstream because it depends upon the entrainment between the centreline of the plume and the top water surface. The deflection reduces as the depth of the discharge pipe increases. Entraining ambient water into the plume region reduces the velocity of the plume, so the velocity above the centreline of the plume would be higher velocity than the lower region [29].

2.5.2 Surface Discharge

In surface discharge the discharge pipe is semi-submerged and the distance between the centreline of the plume and the free surface of the receiving water is equal to the radius of the discharge pipe. Surface discharge is preferred in some cases as the majority of heat transfer to the atmosphere is by evaporation and radiation. The receiving water is less affected by surface discharge as the heated plume remains on the surface, so only a small amount of heat distributes to the layer below the centreline of the plume. Therefore aquatic life can safely pass through the undisturbed space between the bed and plume [29]. This type of discharge is easy to investigate as the discharge pipe is located on the surface and the thermal plume can be observed. In addition the vertical velocity and buoyancy effects are limited.

2.6 Thermal Outfall types

2.6.1 Buoyant Jet

A buoyant jet or forced plume is a flow of water with low density discharged with high initial velocity through an orifice into a receiving water of higher density. The thermal discharge in this case has high kinetic energy and momentum. The buoyant jet flow is fully turbulent whenever its efflux Reynolds number, based on efflux velocity, orifice dimension and fluid kinematic viscosity is sufficiently large [26] .

2.6.2 Pure Plume

The thermal discharge called pure plume occurs when low density water discharges with a low initial velocity through an orifice into higher density of receiving water.

Jets and plumes are classified to four different regions as follows:

Core region

A small region around the discharge outfall in which the temperature and velocity are very high and remain nominally constant.

Entrainment region

In this region the centreline velocity and temperature are decreased significantly, the lateral spread is much greater than vertical spread.

Stable region

In this the vertical entrainment is reduced or ended, the plume depth (thickness) is very small as the heated water spreads on the surface, the temperature remains relatively constant and velocity drops sharply.

Heat loss region

The end of the plume or the discharge may no longer be considered as a plume. The lateral spread is very large and provides a large surface of convective heat transfer. In this region temperature reduces to reach ambient water temperature.

2.7 Equations and factors used

According to the first law of thermodynamics (conservation of energy) energy can neither be created nor destroyed, only can be converted from one form to another. i.e.

change in heat storage = net heat flux = heat energy in - heat energy out

Hence, the net heat flux for a water body can be represented as,

$$q_{net} = q_{sw} + q_{atm} + q_b + q_l + q_h + q_g \tag{1}$$

where,

q_{sw} is short-wave (or solar)radiation,

q_{atm} is downwelling long-wave (or atmospheric)radiation,

q_b is upwelling long-wave (back, or water surface) radiation,

q_l is latent heat flux,

q_h is sensible heat flux,

q_g is conduction between the water and the bed.

2.7.1 Solar radiation

Solar radiation is radiant energy emitted by the sun from a nuclear fusion reaction that creates electromagnetic energy. The magnitude of solar radiation reaching the water's surface depends on the position of the sun, a function of time of day, day of year, and site location, and attenuation of the solar beam due to atmospheric particles and cloud cover. Solar radiation is always positive in sign during the day, zero during night time hours.

$$q_{sw} = H_0\, a_\tau(1-R_s)C_2 \tag{2}$$

H_0=Amount of solar radiation reaching the earth's outer atmosphere

R_s=Albedo or Reflection coefficient

a_τ= Atmospheric attenuation

C_a=Fraction of solar radiation not absorbed by clouds

$$C_a = (1-0.65C_L^2) \tag{3}$$

C_L=Fraction of sky covered by clouds

Extraterrestrial radiation:

The flux of short-wave radiation reaching the earth's outer atmosphere may be estimated by,

$$H_0 = \frac{H_{sc}}{r^2}\left\{sin\varphi sin\delta + \frac{12}{\pi} cos\varphi cos\delta\left[sin(h_e) - sin(h_b)\right]\right\}\Gamma \tag{4}$$

φ= Latitude of the local meridian

δ= Declination in radians

H_{SC}=Solar constant

h_b=Solar angle at the beginning of the time period over which H_0 is calculated

h_e=Solar hour angle at the end of the time period

r=Relative distance between the earth and sun

Γ= Correction factor for diurnal radiation flux

2.7.2 Long wave radiation (q_{atm} and q_b)

Long wave radiation is specified as one of two types: downwelling radiation (q_{atm}) is emitted by the atmosphere, upwelling radiation (q_b) is emitted by the water surface. Longwave radiation is typically calculated using the general form of the Stefan-Boltzmann equation,

$$q_{lw} = \varepsilon\sigma T^4 \tag{5}$$

where

ε is emissivity,

σ is Stefan-Boltzmann constant (5.67x10^{-8} Wm^{-2}K^{-4}),

T is temperature (K)

Downwelling Longwave Radiation

$$q_{atm} = 0.97\ \sigma\alpha_0(1 + 0.17C_L){T_a}^6 \tag{6}$$

where T_a is air temperature (K),

α_0 is a proportionality constant = 0.937x10^{-5}

Upwelling Longwave Radiation

$$q_b = -\ 0.97\ \sigma(T_w + 273.16)^4 \tag{7}$$

where T_w is water temperature (^{o}C)

2.7.3 Latent heat flux (q_L)

Latent heat for various phases of water changes as a function of temperature. The latent heat flux may be represented as equation below,

$$q_l = \rho_w L_v E_r \tag{8}$$

where E_r is evaporation rate given by,

$$E_r = [e_s T_\omega - e_a]f(U) \tag{9}$$

f(U)= Wind function

$T_W e_S$= Saturated vapour pressure computed at water temperature

E_r=Evaporation rate

e_a= Measured vapour pressure

ρ_w is density of the water (kg m^{-3}) being evaporated

L_v is latent heat of vaporization (J kg^{-1})

2.7.4 Sensible heat flux (q_h)

It is the type of heat generated through the molecular transfer between the air and water surface. It depends on the Temperature gradient in the vertical direction.

$$q_h = \rho_w L_v f(U) C_B \frac{P}{P_{ref}(T_a - T_w)} \tag{10}$$

where

f(U) is the wind function (mb^{-1} m s^{-1})

ρ_w is density of water,

L_v is latent heat of vaporization (J kg^{-1}).

The Bowen Ratio describes the relationship between heat and vapour transport which is found to be valid over varying conditions.

The Bowen Ratio is described by the equation,

$$B = \frac{q_h}{q_L} = C_B \frac{P}{P_{ref}} \left(\frac{T_w - T_a}{e_s(T_w) - e_a} \right) \tag{11}$$

P_{ref}= Reference pressure at mean sea level

T_w= Water surface temperature

T_a= Air temperature

e_a= Vapour pressure of the air

Q_l=Latent heat of vaporization

$$Q_l = \rho_w L_v E_r$$

p_w= Density of the water

L_v= Latent heat of vaporization

E_r= Evaporation rate

Net heat exchange across the water surface

$$\Delta H = \{(H_s - H_{sr}) + (H_a - H_{ar})\} - (H_{br} \pm H_c \pm H_e) \tag{12}$$

H_s=Shortwave solar radiation

H_{sr}=Reflected shortwave solar radiation

H_a=Longwave atmospheric radiation

H_{ar}=Reflected longwave atmospheric radiation

H_{br}=Longwave radiation from water

H_c=Conductive heat transfer

H_e=Evaporative heat transfer

2.7.5 Ground heat conduction

Heat flux through soil is governed by thermal conductivity, heat capacity and thermal diffusivity. It highly depends on the moisture content of the soil.

$$q_b = -k_b \frac{\partial T_b}{\partial zz} = 0 \tag{13}$$

where

T_b is streambed temperature

z is vertical distance into the streambed.

k_b is thermal conductivity (W $m^{-1}K^{-1}$) of bed material, a function of thermal diffusivity and heat capacity of the bed material.

2.7.6 Heat exchange for lake

$$\rho c_p V \frac{\partial T}{\partial t} = \rho C_p Q_{in} T_{in} - C_p QT - kA_S(T - T_e) \tag{14}$$

Assumption: Completely mixed lake

One dimensional equation for transport of heat in a reservoir may be written as,

$$Q_z \Delta_z \frac{\partial t}{\partial z} + v \frac{\partial t}{\partial z} = \Delta_z A_z \frac{\partial}{\partial z}\left(D_z \frac{\partial T}{\partial z}\right) + \frac{q}{\rho_\omega c_s} \Delta x \Delta y \tag{15}$$

T=Average temperature

Q_Z=Vertical advection

A_Z=Surface layer of particular element normal to direction of flow

D_Z=Effective diffusion coefficient

p_w=Density of water

q=net heat flux at the air water interface for the surface layer, solar radiation

C_S=Specific heat of water

Heat in a river is most formally described by the 3-D advection diffusion equation

$$\frac{\partial T}{\partial t} + u_x \frac{\partial T}{\partial x} + u_y \frac{\partial T}{\partial y} + u_z \frac{\partial T}{\partial z} = \frac{\partial}{\partial x}\left(D_x \frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(D_y \frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(D_z \frac{\partial T}{\partial z}\right) + \frac{q_{net} A}{\rho C_S V} \tag{16}$$

$$q_{net} = k[(T_e) - T]$$

K=Heat exchange coefficient

T_e=Equilibrium temperature

2.7.7 River

$$U\frac{dT}{dx} = -\frac{k}{\rho c_p H}(T - T_b) - k_{R(T-T_b)} \qquad (17)$$

T_b=Average background water temperature in the absence of the heated discharge

$$T = [(T_0 - T_b)]e^{\frac{-k_R x}{U}} + T_b \qquad (18)$$

T_0=Resulting temperature at the outfall after mixing with the upstream river temperature

2.8 Different methods to reduce effluent temperature

For meeting the cooling requirements of turbine condenser, water is drawn from large water bodies such as sea, lake or river using intake structures. While passing through the condenser, temperature of the condenser cooling water rises and the heat is discharged to the ultimate heat sink. This is achieved by either discharging the hot water to the sea /lake /rivers though the outfall system or by rejecting heat to the open atmosphere through cooling tower. It is ensured that with the given arrangement of intake and outfall structures, the temperature difference between the two legs at specified locations are within the limits stipulated by the State /Central Pollution Control Board or any other appropriate authority. Appropriate numerical/experimental model studies are generally conducted for this purpose.

There are occasions when it is not possible or practicable to reject heat to a natural water body, especially when no land is available for any type of cooling pond or reservoir. In these cases it might simply be that water in sufficient quantity and quality cannot be found in the neighbourhood of the site. On the other hand, it might be that local natural water bodies could not accept the thermal load without causing offence to water quality regulations, or that severe ecological consequences would result. In such cases, one of the following cooling devices must be adopted for a nuclear plant

(a) Cooling towers

(b) Spray ponds or canals

(c) Mixed system

(d) Hybrid cooling towers

(d) Cooling ponds

(e) Spray towers

Cooling towers

In the cooling tower the warmed water is sprayed downward, and air is blown upward with a fan. As the warm water droplets contact the air, some of the water droplets evaporate, and the air absorbs the heat released from this evaporation thereby lowering the temperature of the remaining water. This cooling effect of the remaining water is called the latent heat of evaporation. During this process, some water is lost to the air from evaporation and some water is lost by the misting effect (called “drift”) into the air. These are very effective cooling devices.

In most cases cooling towers of one type or another are used. There are many types available and the terminology applied stems from basic differences in design or operation which serve to categorize them. A tower may be either 'wet' (evaporative) or 'dry' depending on whether or not the cooling water is exposed to the air; 'mechanical draught' o r 'natural draught' depending on whether or not fans are employed for inducing air movement; ' cross flow' or 'counter flow depending upon or vertical. In mechanical draught towers air flow may be either' forced draught' or 'induced draught'' depending on whether the fans push the air through from the bottom or side or pull it through from the top of the packing.

Spray ponds

A spray pond [27, 28] is a pond with a spray system located about 2 to 3 m above a large natural or artificial reservoir. Although more compact than a cooling pond it has some disadvantages:

a) Limited performance available because the contact time of the
b) sprayed water-with-the-air before reaching the surface of the ponds comparatively limited;
c) A nuisance is created by the high water loss at certain time of the year during high winds, activating corrosion through carry-over of droplets;
d) Since they are open to the atmosphere, ponds collect considerable quantities of foreign matter which pollutes the water.

But spraying reduces considerably the pond area required : by a factor between 30 and 100.

Mixed systems

Mixed systems are required:

a) As series devices to achieve the thermal quality standards before direct discharges can be made to the water body. Examples of these are the use of spray ponds, cooling ponds or cooling towers prior to such discharge.

b) As parallel stand-by devices where large seasonal changes occur in air temperature and humidity and in water temperature and availability.

Extreme annual range of air temperature may require dry cooling in winter and wet cooling in summer

Hybrid cooling towers

It can be operated either as evaporative or as dry are now being conceived to satisfy this requirement. Where river flow or lake storage is poor in certain seasons and undergoes large seasonal variations cooling towers may be used instead of direct cooling in such seasons.

Cooling ponds

A cooling pond is a shallow reservoir designed to receive warm water and discharge cool water, relying on evaporative and radiative heat loss. Cooling ponds are typically lined with an impermeable material that is resistant to ultraviolet radiation and punctures, and that can withstand the temperatures to which it is exposed. This method is used where the temperature difference is high and land cost is low. This method is easy to design and operate.

Spray towers

The heated water is repeatedly sprayed into the atmosphere from a lined pond, exposing millions of droplets to the air which absorbs some of the moisture, transferring heat out of the remaining water. The operative physical cooling principle is the same as in a cooling tower or cooling pond. Spray cooling uses a pump and motor, manifold, nozzles, and a floating platform. This approach requires more space than a cooling tower, but less land than a cooling pond. This method is more expensive than cooling ponds and less expensive than cooling towers.

2.9 Site Selection

An important stage in the development of a thermal or nuclear power project is the selection of a suitable site to establish the site-related design inputs for the Power Plant (PP). The selection of suitable site is the result of a process in which the costs are minimized. It is also to ensure adequate protection of site personnel, the public and the environment from the impacts of the construction and operation of power plant. The key elements for selection of site for a Thermal or Nuclear Power Station include:

- The availability of suitable and adequate land with least R & R issues
- Fuel availability and its transportation from the source of availability

- Water availability within a reasonable distance
- Road and Railway access
- Acceptability from the Environmental consideration
- Availability of infrastructural facilities
- Rehabilitation and Resettlement issues (R&R)
- Proximity to Grid for Evacuation of Power

Chapter 3

DATA REQUIREMENT

3.1 Introduction

Data is an important aspect of any type of study. It is the base of any civil work on which the very existence of a feasible solution lies. Any inaccurate data may lead to invalid result and affect result of the study. There are various types of data such as waves, tides, currents and meteorological data like wind, ambient water temperature, relative humidity, cloud cover, solar insolation etc. which are required to take into account for thermal dispersion studies.

The primary purpose of the intake system of a power plant is to provide a reliable source of water in the proper quantity and at the proper quality and temperature to ensure satisfactory operation of the plant. At first glance, supplying water from a water body such as river, canal, reservoir, ocean etc. to a power plant appears to be a relatively simple task. A water body particularly river and ocean is a dynamic entity which is constantly in motion and, therefore, constantly changing the bank or shoreline and the bottom profile. These changes are the result of the action of hydrodynamic phenomena such as currents and waves which are capable of moving hundreds of cubic meters of sand and sediments. The ocean or river is extremely powerful and can create devastating forces in short periods of time. Seawater also acts slowly to cause the incipient corrosion of submerged structures. In addition, the ocean is alive with marine organisms which can rapidly attack or foul submerged objects. All of these factors combine to complicate the installation of equipment in the water body and make the design of intakes a task which requires careful attention and planning. Because the design of intake systems superficially appears simple, often the data used is insufficient to evaluate the many parameters that can adversely affect the performance of a intake properly. For this reason, problems often develop in the operation of intake systems. Consequently, problems with seawater intakes and corrosion are the two primary causes for unscheduled downtime in power plants. The following sections outline the data required for deciding the locations of intake-outfall of thermal /nuclear power plants.

Environmental data to be considered in relation to nuclear power plant and nuclear fuel projects include land, air, noise, water, marine, biological socio economic, health, and

background radioactivity. In the case of coastal plants, data is to be collected as part of baseline study, along with oceanographic data covering the following parameters:
Tides, Waves (wind waves and swells), Storm surges, Currents, Salinity, Sea water, temperature, Suspended load and their profile upto the proposed water, Seabed bathymetry discharge point

Meteorological data covering Wind speed and direction, Rainfall, Relative humidity, Temperature, Barometric pressures and History of cyclones /floods data for at least a 10-year period should be presented from the nearest meteorological station, except for the history of cyclones and floods for which 50-year data is required [39]. Further the details about baseline and meteorological data are also required as detailed below.

3.2. Site Conditions and baseline data

The objective of an intake system is to provide a reliable source of fresh water within a proper temperature range which is free from contaminants. A proper assessment of site conditions for water intake is of fundamental importance for meeting this objective [30]. To evaluate a site for intake properly, the factors that must be considered include physical site characteristics and meteorological and oceanographic data. In addition, potential sources of contamination such as fouling by organisms, oil spills, or other pollution should be evaluated.

3.2.1 Water Depth

When considering the location of a power plant near a water body along the bank or shoreline, it is important to remember that the water body such as river or sea is in constant motion and is constantly changing. Water levels vary during monsoon in a river and on a daily basis in sea as tide levels change.

Particularly, in sea a structure well above the waterline in the morning may be submerged 12 hour later. In addition, forces caused by waves and currents are constantly at work modifying the shoreline and the profile of the sea floor nearshore. These forces can result in changes of several meters in elevation in just a few weeks time. For these reasons, it must be realized that water depth is not a set figure; instead it is a variable that changes with both the time of day and the time of year and is often expressed in relation to certain reference datum [39]. In the case of sea, consideration is generally given to the effects of water depth on intake performance characteristics. The surf zone is the area in which waves approaching the shoreline break. Breaking waves create a great deal of turbulence. The churning motion of breaking waves lifts particles from the bottom into suspension and causes water in the surf zone to be more turbid and to have higher levels of suspended solids. For this reason, it is not advisable to take seawater directly from the surf zone. Breaking waves also exert a tremendous amount of force. Objects installed near the surface in the surf zone could be subject to the direct impact of waves. The forces associated with wave movement decrease rapidly with depth. Therefore, by locating a structure at a sufficient depth below the water surface, the wave forces on the structure can be minimized. Water temperature also varies with depth. The air-water interface is a heat transfer surface. The air heats up more rapidly during the day and cools off more quickly at night. Water, with its higher heat capacity, tends to change temperatures more slowly than air. Figure 8 illustrates the heat transfer which occurs at the air water interface.

The primary source of heat input is direct radiation from the sun. High percentages of this heat are returned to the atmosphere in the form of evaporation and long-wave radiation. The incoming radiation is attenuated rapidly by the water with 50 per cent being absorbed in the top centimetre and 90 per cent being absorbed in the top 40 m. In the deep oceans, this can result in thermoclines of 20 to 30°C between the surface and the bottom [39]. Because of the rapid attenuation of radiant energy with depth, thermoclines of several degrees centigrade can exist between the surface and a depth of several meters. The magnitude of this surface thermocline is affected by the degree of mixing caused by waves. In shallow, protected bays, where there is limited heat transfer to cooler deeper waters and where wave-induced mixing is restricted, surface water temperatures may be several degrees centigrade higher than in open water.

It is important to keep water temperatures within a proper designed range for seawater intake. To reduce temperature fluctuations which can be experienced in stratified surface layers, it is preferred that seawater be obtained from a deeper layer. One of the first considerations then, concerning the location of an intake for a desalination plant, is the proximity to the shore of a location deep enough to obtain cooler, less turbid water. Either a pipe- or channel-type intake can then be selected, depending on seabed conditions, to bring the water from this location to the plant. NPP requires reliable sources of water for steam condensation, service water, emergency core cooling system and other functions [42].

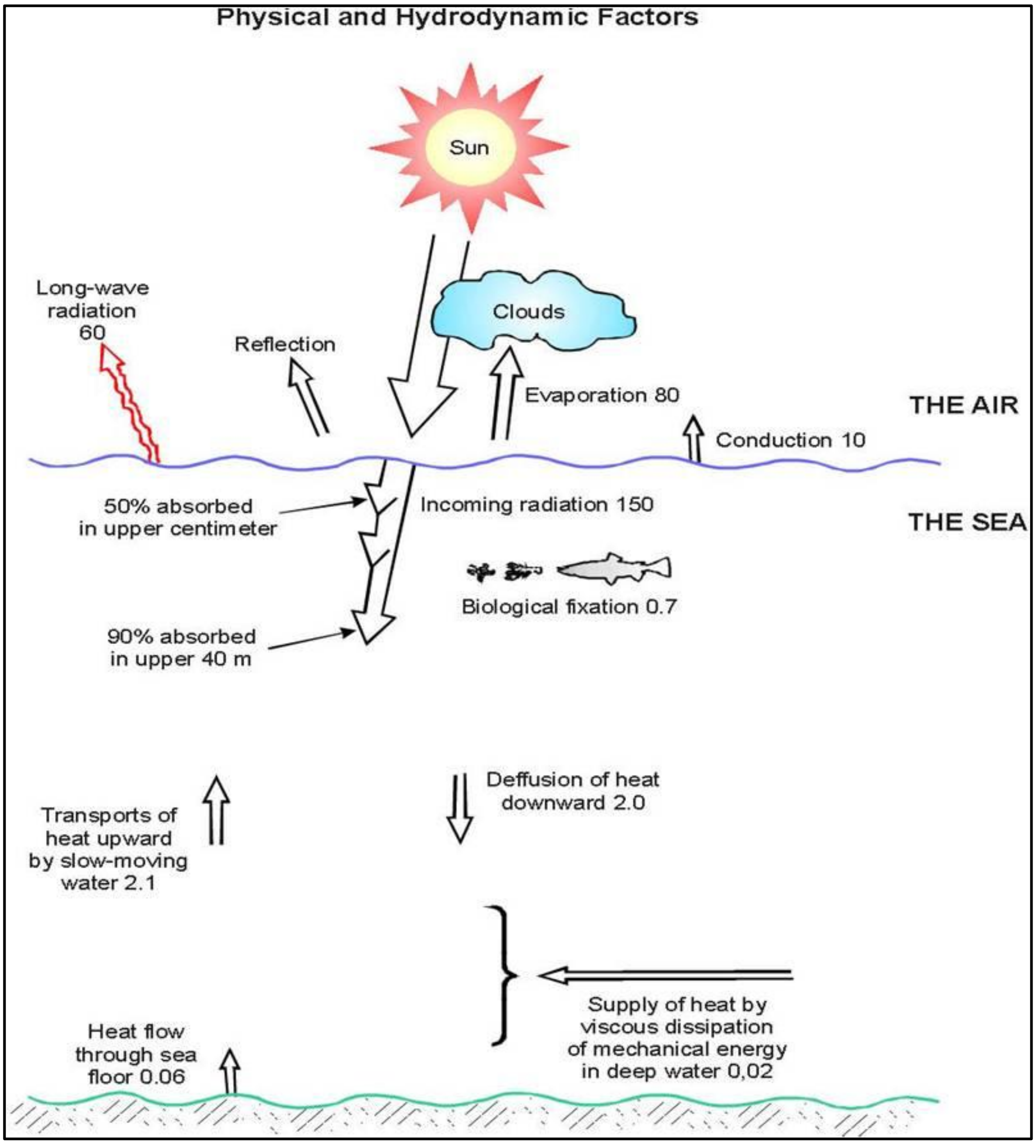


Fig. 8 Heat transfer mechanism of the oceans

Where water is in short supply, the recirculation of the hot cooling water through cooling towers, artificial pond, or impoundments has been practised. The availability of essential water during periods of low flow or low water level is an important initial consideration for identifying potential sites on rivers, small shallow lakes or along coastlines. Both the frequency and duration of low flow or low-level periods should be determined from the historical record and, if the cooling water is to be drawn from impoundments, they should be determined from projected operating practices.

In determining the adequacy of water supply, the following characteristic of water supply should be taken into account, in relation to proposed NPPs design.

a) Water supply flow rate (Make-up Flow Rate for Close Cycle System and Cooling Water Flow Rate for Once-through System)

b) Maximum consumption of water supply

c) Monthly average consumption of water supply

For lakes, the supply capacity would be evaluated considering the capacity and water level of the lake, as well as historic low levels and refill (inflow) rates, together with the potential for conflict with lake usage, such as recreation. A site with minimal impact to water quality will be most favourable than a site which gives maximum impact to water quality.

3.2.2 Seabed Conditions

The seabed conditions will be one of the primary factors in determining the type of seawater intake structure for a particular location. If the intake is to draw water from the open coastline and not from a sheltered lagoon, the two most common types of intakes are the pipe and the channel intake. In the pipe-type intake, a pipeline is run from the shoreline out past the surf zone. For sandy bottoms, a trench is usually dredged, the pipe laid, and the trench backfilled. For bottoms composed primarily of rock or coral, dredging and backfilling would be difficult and expensive. Pipelines are typically buried for anchorage and protection from wave forces and currents [30]. Other methods of anchorage besides trenching and backfilling have been employed, including multi helix anchors, concrete saddles, engineered backfill, and grouted neoprene impregnated nylon bags or pillows filled with grout. There are other problems associated with laying pipelines on a rocky bottom. Pipes laid in uneven terrain will only be supported at the high spots. This bridging between support points results in stresses that must be carried by the pipe. In addition, the danger of abrasion of the pipeline is increased with rock bottoms. For these reasons, rock bottoms are not conducive to the installation of a pipe-type intake.

A channel type intake can be constructed either by installing rocks to form walls along each side of the channel or by dredging. In sandy bottoms there is a tendency for the movement of sand due to littoral transport. This is discussed in more detail in a later section. This littoral transport can either act to erode sand from the channel walls or result in sand being deposited in the channel. This deposition of sand could result in blockage of the channel requiring dredging to keep the channel open. In addition, wave action in the channel would result in sand and silt being placed in suspension causing higher turbidity. The above problems can all seriously affect the operation of a channel-type intake for a sandy bottom. A detailed investigation of sediment gradation, settling rates, littoral transport patterns, currents, and wave activity is required for the design of a channel type intake in silty or sandy bottoms. For rocky bottoms, the problems of littoral transport, erosion, and suspension of particles due to wave activity are not significant considerations in designing channel type intakes. The bottom is generally more stable and problems of shifting sand and sediments should not occur [30]. With properly selected materials for channel protection, erosion of channel walls should not be a problem. In addition, wave action will not result in the stirring up of sediments where rock bottoms are found. A channel-type intake installed on a rock bottom would not be encumbered by the sediment transport problems associated with a sandy bottom. The design of a channel-type intake on rock bottoms would primarily be concerned with resisting wave forces and eliminating debris. It can be seen that a sandy bottom favours the installation of a pipe-type intake system while a rocky bottom tends to favour a channel-type intake.

3.2.3 Waves and Currents

The oceans are extremely dynamic systems constantly in motion as a result of external forces, such as wind and internal forces, such as temperature and salinity gradients. The rotation of the Earth results in oceanographic currents and wind forces cause waves and also affect the currents. Seismic forces in the Earth may also result in disruptive waves in the form of tsunamis and seiches.

In many situations, waves are the most important parameter, which influences the coastal phenomena and the related design considerations. Waves in the generation area are of variable heights and frequencies and propagate in all directions. These are known as sea waves. However, when waves leave the generation area and travel towards the coast they get segregated according to frequency due to dispersion of waves, which are known as swell waves. The wind energy in the offshore region is transported to the coast in the form of waves which constantly agitate the coastal region. The waves cause dynamic impact on coastal structures, which they must withstand. At coast, the waves result in movement of sand along the shore causing erosion or accretion of shoreline. The waves stir up the sediments at the bed and bring them into suspension, which are transported by currents, which may lead to siltation in harbours and approaches. Therefore, understanding mechanism of waves and the information on wave climate is important to coastal engineers. Wave climate refers to the general condition of sea state at a particular location. The important parameters in wave climate are significant wave height (Hs), peak wave period (Tp), average period (Tz), and mean wave direction [46, 51]. It is normally expressed in the form of wave rose diagram. Moreover, the construction of a coastal structure requires additional data on extreme design wave conditions.

Ocean Current is a continuous, directed movement of seawater generated by forces acting upon this mean flow such as breaking waves, wind, the Coriolis Effect, temperature and salinity differences. Ocean currents are primarily continuous, horizontal and directional sea water movements generated by a number of forces like wind, the Coriolis effect, breaking waves, tides, freshets, oceanic circulation, cabbeling, temperature and salinity differences etc. acting upon the water. Measurements of magnitude and direction of currents are essential to estimate accretion/erosion, siltation patterns. Tidal currents are generated due to periodic rise and fall of water level caused by tides. The tidal currents play an important role in the coastal processes especially in the estuaries and near the river mouths [46, 51]. Near-shore currents are further classified as Long-shore currents, rip currents, onshore – offshore currents. All these currents pose different kinds of problems pertaining to different structures depending upon the location of the structure. The measurement of current is normally carried out at surface, mid-depth and at bottom depending upon the application. Acoustics Doppler current profilers (ADCP) have been widely used these days for current measurements.

3.2.4 Tides

In generalized way, tides are defined as the periodic rise and fall of the water level in the sea. They are produced by the attraction force of sun, moon, earth and other celestial bodies. The rising of water level is called as flood tide while the lowering is called as ebb tide. When the moon, sun and earth are in same line then tides produced are called as spring tides, where as when moon, sun are at quadrature to earth then tides are neap tides. The structures constructed in coastal areas such as breakwaters, seawalls, groins, power plants and other onshore coastal structures are exposed to tidal variations [46, 51]. Hence, knowledge of tidal levels is essential to decide the crest level of the structures so that the amplified wave action with the highest high water level does not overtop the structure and to finalize the location of the structures considering tidal variation in vertical direction. The tidal levels always need to be correlated with a standard datum such as the Chart Datumor (CD) GTS bench mark.

3.2.5 Safe Grade Elevation

Safe grade level is the plinth level of any plant high enough not likely to get affected or submerged by surrounding water level during extreme conditions of floods, tides, waves, storm surge, inundation, cloud burst , tsunami or any other calamities [30]. For each site there are different factors like topography, meteorology, hydrology of the region, site specific features etc. which influence the flooding pattern or inundation under different flooding conditions. Water level fluctuation due to the astronomical tides which depend upon the location, Latitude, Longitude , Phase of the moon (lunar cycle) causing the tide, waves, wave uprush, Tsunami etc are required to be taken into account for finalization of safe grade elevation for a coastal structure.

3.3 Meteorological parameters

3.3.1 Wind

Ocean environment is always associated with wind. In ocean environment, any structure projected above water level is subjected to wind load. Winds are also responsible for storm water level set up. Such storms can cause even submerge of large areas of land. Besides generating waves, wind also produces current and water level set up and set down. Position of coastline with respect to wind direction governs current field. It is observed that water does not move in the direction of wind. Once water particle starts moving due to wind, friction force and Coriolis force start working on that and particles starts moving in a direction making an angle with wind direction [51]. The wind data is normally available from the observations and daily weather reports published by the India Meteorological Department. The wind rose diagram gives information about percentage of occurrence of wind speed, direction as well as percentage of calm period.

3.3.2 Water Temperature

Many physical properties of materials including the phase (solid, liquid, gaseous or plasma), density, solubility, vapour pressure, and electrical conductivity depend on the temperature. Temperature also plays an important role in determining the rate and extent to which chemical reactions occur. Temperatures of higher degrees can result in harmful reactions with serious consequences [51]. Temperature also controls the type and quantity of thermal radiation emitted from a surface. Thermal changes can have pronounced effect on water quality [15].

3.3.3 Relative Humidity

Relative humidity (RH) is the ratio of the partial pressure of water vapour to the equilibrium vapour pressure of water at a given temperature. Relative humidity depends on temperature and the pressure of the system. The same amount of water vapour results in higher relative humidity in cool air than warm air. High moisture content can affect the overall productivity of the power plant. Relative humidity affects the evaporation rate directly. So it is a very important factor in modelling the heat exchange phenomenon of thermal simulations.

3.3.4 Cloud Cover

Cloud cover is the most important meteorological factor determining the amount of solar radiation reaching the Earth's surface. It is the fraction of the sky obscured by clouds when observed from a particular location. The cloud cover is correlated to the sunshine duration as the least cloudy locales are the sunniest ones while the cloudiest areas are the least sunny places. Clouds efficiently reflect light to space and thus contribute to the cooling of the planet. Clouds both reflect incoming sunlight and inhibit the radiation of heat radiation from the surface, thereby affecting both sides of the global energy balance equation.

3.3.5 Solar Insolation

Solar Insolation is the power per unit area, received from the sun in the form of electromagnetic radiation. It is measured at the earth’s surface after atmospheric absorption, scattering, reflection and shading. It is a function of distance from the sun, solar cycle, cross-cyclic changes, tilt of the measuring surface, height of the sun above the horizon and atmospheric conditions. Insolation levels, measured in W/m^2 change throughout the year; lowest in the winter and highest in summer. It is the highest when the Sun is directly overhead in an area [51]. So it is highest in the equatorial regions and lowest in Polar Regions. The net solar insolation is incorporated in the equations given in Chapter 2.

3.4 Importance of accuracy of the Data

As the model relies heavily on the data of climatic factors, it is necessary to take due care in the field collection of data. Further, continuous monitoring of the water body temperature and climatic data is helpful in tuning of the developed model. The model can successfully predict the results only and only if the data which is being used is precise and accurate. Since the data plays an important role in calibration and validation of the model and also impacts various other parameters directly or indirectly, sensitivity analysis if the data is absolutely necessary.

Chapter 4

MODELLING TECHNIQUES

4.1 Introduction

Before one can assess the transitional impact of thermal releases on a particular receiving body of water, it is necessary to determine the areal extent and temporal behaviour of the temperature changes within the body of water induced by the heated discharges. The proper design of discharge structures for once-through systems is the most important factor in determining the magnitude, extent, and distribution of thermal effects in the receiving water bodies. The increased size of nuclear units and a growing concern with the environmental effects of water temperature changes have combined to limit the number of sites where once-through condenser cooling can be used. However, by combining good engineering design with proper assessment of thermal effects, it is believed that once-through cooling can remain a viable mechanism for heat dissipation in major rivers, reservoirs, large lakes, and coastal waters. Heat dissipation from the receiving water surface will ultimately return the water to its natural temperature state within a certain distance from the point of heat discharge. In general, the flow-rate in a river or reservoir must be at least three times the discharge flow from the power plant and complete mixing of the heated water with the river (or reservoir) flow is required within a zone of not greater than 500 metres from the point of release. This distance depends on the amount of mixing or dilution between the heated condenser water discharge and the receiving water. The transfer of heat from a water surface to the atmosphere occurs through the combined mechanisms of evaporation, radiation, advection and conduction. Kolflat [29] has provided typical values for each of these heat dissipation mechanisms for open water surfaces such as lakes, ponds, rivers, reservoirs, and estuaries. These values are given as: evaporation 40%, radiation 30%, conduction 25%, and advection 5%. If cooling towers are used, over 75% of the heat is transferred by evaporation in the 'wet' type in the summer, and in dry cooling towers 100% of the heat is dissipated by conduction and convection. This extreme type of non-mixing discharge is usually prohibited by prescribing natural temperature differentials in the receiving water. This can be done by a design which provides for partial or complete mixing of the heated discharge with the available flow past the plant site. In this design, the condenser cooling water is conducted through a diffuser pipe or tunnel and discharged through nozzles or ports near the bottom of the water-way. Entrainment of surrounding water into the high-velocity jets produces a rapid dilution. This type of discharge device provides the most rapid temperature reduction within the smallest area. On the other hand, a maximum amount of heat is stored in the water since surface heat dissipation is relatively slow at the very low temperature rises permitted.

The mixing process is generally investigated in the near field or the far field. The near field is defined as the region in which the mixing is controlled by the initial discharge momentum and buoyancy flux as well as the outfall geometry [25]. Effluent mixing in the far-field is mainly caused by the ambient flow advection and diffusion. This is also known as passive diffusion and its mixing process is controlled by the turbulence characteristics of the ambient flow [22, 23]. The near field and the far field mixing processes are different with respect to their length and time scales. Length scale addresses the largest possible eddy size that causes the mixing. These eddies are sources of momentum, mass and energy transfer in and out of a control volume. For example, the transverse eddies that rotate horizontally, about a vertical axis causes lateral mixing in a river [18]. The intermediate field definition also has been used in some studies as the region downstream of the outfall in which the mixing process is still affected by the initial momentum and buoyancy characteristics. Due to the large width to depth ratio, the vertical mixing usually occurs close to the outfall in water bodies and the near field region is considered from the discharge point up to the point of full vertical mixing. The intermediate field in water bodies is defined between the full vertical mixing to the point where full transverse mixing occurs and the far field region begins after this point [1]. The full transverse mixing length in water bodies is defined as the longitudinal distance required to achieve complete mixing (equal concentration) across the channel [48]. The full mixing condition is assumed when the ratio of the minimum to the maximum concentration ($Pm=Smin/Smax$) reaches the certain limit, normally 0.98, 0.95 or 0.9 [48].

The outfall systems can be generally categorized as single port, multiport and surface outfalls [44]. They are also different with respect to the discharge capacity, construction cost and dilution potential. Surface outfalls are of the most popular outfall systems due to their low cost and large discharge volume capacity. Effluent outflows generally have different velocity and density from the receiving water. Therefore, the effluent is named as jet in the case of momentum driving force or plume when it is driven as a result of buoyancy. Effluents are generally called buoyant jet or forced plume in the literature if they are influenced by both momentum and buoyancy effects [27]. The surface jet flows can be divided into three principal regions as the Zone of Flow Establishment (ZFE), the near field and the far field [33]. The ZFE is the closest region to the outfall. The effluent has uniform "top hat" shaped velocity and temperature profiles at the discharge location. The velocity difference between the ZFE jet and the ambient water produce shear and consequently turbulence around the jet. The ambient water entrains the ZFE and decreases the width of this region with distance from the outfall. The extent of ZFE is defined from the outfall location to the point that the jet centreline velocity or temperature starts to decay from the discharge value. The length of the ZFE for temperature is less than velocity due to the higher turbulent diffusion coefficient [33]. The next region is the near-field, also known as the Zone of Established Flow (ZEF), in which the mixing is affected by the initial volume, momentum and buoyancy fluxes. Velocity, salinity and temperature all have a self-similar Gaussian shape profile in this region. Surface outfall flow regimes are generally classified as: free jets, shoreline-attached jets, wall jets, and upstream intruding plumes [27, 28]. The momentum difference between jet region and the ambient water in the near field generates shear and consequently turbulence which is the main mixing mechanism within the initial portion of the near-field. Jet like mixing in this region increases the plume thickness and leads to full vertical mixing in shallow ambient water. The density gradient between the buoyant-jet and the ambient fluid also produces buoyancy forces due to unbalanced hydrostatic pressure, which also enhances the mixing process. This buoyant mixing mechanism is more significant further from the high momentum region near the outfall. In a stratified flow the buoyant plume will reach a level of equal density, whereupon it undergoes lateral buoyant spreading and reduced vertical mixing. The "Far field" is the region where the plume is transported by the ambient current and mixed by the background turbulent diffusion. Therefore this region is also known as the passive diffusion region. The far field mixing time scale varies from the hours to days and its length scale is thousands of meters [40]. The dilution rate in this region is much slower than the near field. The unsteady nature of tidal currents as well as wind generated waves influence mixing in this region, which makes the coastal mixing process complicated in the far field. Far field mixing can be modelled using Eulerian or Lagrangian approaches [47].

4.2 Modelling Methods

Various methods exist to determine the size of a temperature field from a thermal discharge. Because of the difficulties in simulating actual turbulent jets into real environments, all of these methods are approximate. No exact method presently exists. They, however, can be used to predict temperatures with reasonable accuracy. Each method has its advantages and disadvantages. The problem is to know the limits of each method and when to use which one.

4.2.1 Physical modelling

Physical modelling is a technique where a scale model of the receiving water and discharge system is constructed to carry out measurements that can be used to estimate the response in the actual discharge. Dynamic, kinematic and geometric similarity laws are used to determine size, shape and operating conditions of the model. They are then used to predict the temperatures in the actual discharge from measured values in the model. Dynamic similarity requires that various dimensionless parameters such as Reynolds number, Froude number and Nusselt number be the same for both the physical model and full sized discharge and environment. Geometric similarity requires that all dimensions in the model be the same as the values in the full sized project multiplied by a fixed scale ratio. Further, scale effects arise in physical modelling due to force ratios which are not identical between a model and its prototype and result in deviations between the up-scaled model and prototype observations. The advantage of physical models is the ability to include time dependency and the complete geometry of the environment and discharge. The disadvantages are the high cost and the inability to look at variations with ease.

Because various lengths are contained within the dynamic similarity parameters, it is often impossible to have all dimensionless parameters the same in both the model the actual discharge and have the model smaller than the actual discharge. As a result, it is common practice to ignore some of the minor parameters. For example, with submerged, buoyant discharges, similarity is often based on the Densimetric Froude number. The Reynolds number is ignored as long as it is large enough to insure a turbulent jet. It is then assumed that as long as the densimetric Froude number is the same in the model as in the actual discharge, temperature ratios determined in the model will be the same as in the geometrically similar full scale discharge. This assumes that ignoring Reynolds number and other minor parameters will have no effect on the relationship between values in the model and the actual discharge. Mostly, turbulence is not the same in models as in full sized discharges and occasionally errors occur due to the scale effects.

In large shallow lakes, rivers and estuaries, geometric similarity results in very shallow depths in the model. These shallow depths may introduce effects such as surface tension that are not present in the full scale environment. This results in very large models or the use of a skewed model with different horizontal and vertical length scales. This requires special use of similarity relations.

The advantage of physical models is the ability to include time dependency and the complete geometry of the environment and discharge. This is very important when transients or geometry must be accounted for. The disadvantages are the high cost and the inability to look at variations with ease. It often takes weeks or months to set up a single new case. Presently, not many hydraulic laboratories and universities around the world have facilities for physical modelling.

4.2.2 Empirical Models

Empirical modelling is where laboratory or field data are used to generate algebraic equations that can be used to predict plume development on similar discharges. As in physical modelling, the major variables in the problem are arranged into dimensionless groups or into length scale parameters. These parameters are plotted and regression analyses are used to determine equations the best describe the results. These equations are then used to predict the response of any discharge that falls into the category as the measured data.

The advantage of empirical methods is the ability to include the effects of complex geometries in the equations and that the equations can easily be evaluated. Once the equations are known, they can often be evaluated with a hand calculator. Usually they are coupled to a more complex interface that automatically selects the proper equations. The disadvantages are that an infinite number of equations would need to be developed to cover all possible types of discharges and environments and have continuous predictions when passing from one region to the next. This would require measurements in every possible environment and discharge configuration. Since this is not possible, a finite number of classes are usually developed to cover a range of variables. Discontinuities can exist when going from one class to the next.

The most widely used empirical model is the CORMIX model [25]. Although it is listed under “Empirical Models”, CORMIX also has a multiple port integral method model called CORJET that is used on simpler configurations. CORMIX is a software system that incorporates an expert system interface with a number of hydraulic models. CORMIX is designed to handle a wide variety of discharge configurations and ambiences including single and multiple port diffusers, submerged and surface discharge, positive and negative buoyant discharges, diffusers with unidirectional, fanned, alternating ports or risers with multiple ports, open oceans, lakes, rivers, and estuaries. It can also consider tidal varying ambiences in a quasi-transient approach. CORMIX is inherently steady state, but it also considers the rate of change of ambient conditions and account for re-entrainment due to tidal reversal. Once all the data entered into the dialog boxes, CORMIX determines a number of length scales. These are used to determine the flow class. Flow class is the system used by CORMIX to specify which type of plume is expected to develop as a result of the discharge configuration and ambient. For example, the flow is jet like or plume like; stratification is sufficient to dominate; current is weak or strong; the receiving water is deep or shallow; the alignment is parallel or perpendicular; discharge is positively buoyant or negatively buoyant; there is coanda bottom attachment or wake attachment; there is an upstream wedge; etc. There is something like sixty six flow classes for submerged discharge along. CORMIX then runs the hydraulic models it has determined are appropriate for this flow class in sequence and patches them together.

4.2.3 Integral methods

In integral methods, plume properties are integrated across the face of the plume, perpendicular to the plume centreline using an appropriate approximation to the actual profile. The three most popular profiles are the “top hat” profile which assumes plume properties are constant across the face of the plume, a statistical bell shaped profile based on a Gaussian curve with maximum values at the centre, and a 3/2 power law profile that approximates the Gaussian curve. Variations from actual profiles are lost in the integration since the result is a function of only the plume centreline values and plume size. Since these profiles deal with values in excess to ambient conditions, integration need only be made from the plume centreline to its edge. Actual geometric boundaries are not included and are only accounted for in a secondary manner. When these integrated profiles are incorporated into the differential equations of motion, as system of ordinary differential equations result that can be solved stepwise along the centreline of the plume.

In order to solve the system of equations described above, an entrainment function must be developed and used that simulates the fluid drawn into the plume by turbulent shear and exchange at the plume boundary. It must also include the fluid forced into the plume by ambient current. This entrainment function is the key to success in integral methods. They have been developed in the past using field and laboratory experiments.

The step-wise integration of the equations yields fluid properties including plume size and location at each integration step along the trajectory of the plume. For Gaussian and 3/2 power law profiles, centreline values of temperature, concentration, and velocity calculated. From the assumed profile and plume size, off centreline values can be determined. For "top hat" profiles, average values are calculated. These can be used with an assumed profile and plume size to determine centreline values.

The advantage of integral models is that they can be evaluated in just a few seconds with present micro computers. They also give a good representation of the plume in the near field where physical boundaries are not important. As a result, a large number of cases can be analyzed within just a few minutes and get good answers in the region where temperature and concentration are the highest. The disadvantage is that they cannot be used where physical boundaries play a significant role in the dispersion process. In addition, transient plumes can only be analyzed as quasi-steady state problems with a series of steady state runs with different ambient conditions are used to simulate transient behaviour. Commonly used integral models are VISJET [12], Visual Plumes [17] etc.

4.2.4 Mathematical models

Mathematical modelling of thermal discharges aids in the prediction of possible effects which may result from changes in physical properties of water due to the extent of the influence of thermal discharges. Modelling also provides a technical basis for developing a discharge structure design which disperses the heat effluent in a manner which will satisfy temperature standards and also estimates the possible recirculation of heated water back into the intake structure.

In general, heated water releases are commonly divided into two broad categories, namely: surface and submerged discharges. In these categories, there are five basic processes which contribute to the dispersion of heat in a large receiving body of water [19]. The first three of these processes such as jet entrainment; turbulent diffusion; and buoyant spreading; contribute primarily to the mixing of the heated and ambient fluids. In contrast to these hydrodynamic processes which merely redistribute the heat in the receiving water, the fourth process, heat transfer to the overlying air, transfers the thermal energy to the atmosphere. The fifth process, interaction of the initial jet momentum and ambient cross current, generally determines the location of the plume temperature field in relation to the outfall structure and receiving water body. None of the mathematical models developed to date has successfully simulated all five processes. When heated water is discharged in to a water body, the resulting temperature field can be divided into two distinct zones or regions, and the majority of the mathematical models developed to date consider only one of these specific flow regions i.e. far-field region [44, 45].

(1) An initial or near-field region in which temperature changes are governed primarily by the geometry and hydrodynamics of the discharge. Mechanisms which affect the temperature reduction in the near-field region are the dilution and entrainment due to the momentum of the discharge jet and the buoyancy effects due to the temperature difference between the discharge and the receiving water. Any 'near-field model' or 'jet model' must simulate the full characteristics between the heated effluent and the ambient receiving fluid. Compliance with water temperature standards is also generally determined in the near-field region.

(2) A far-field region in which the temperature distribution is governed by conditions in the receiving water. The important properties of the receiving water body are natural temperature stratifications, advection, diffusion and dispersion due to tidal currents, wind-driven currents and wave action, and heat dissipation from the water surface. A 'far-field model' generally needs to describe only the motion of the ambient currents while, at the same time, satisfying the basic conservation laws for mass and heat. These models generally use the diffusion concept to describe the rate of effluent mixing, with the assumption that transport due to turbulent velocity fluctuation can be lumped into horizontal and vertical diffusion coefficients. Some attempts have been made to couple the intermediate region between near- and far-field regions into a 'complete-field model'. With this region there is a transition from inertia dominated flow to ambient turbulence and gravity-dominated flows. Ideally, the coupling between the hydrodynamic equations should be considered in this intermediate flow regime and much work remains to be done to adequately understand this area. It should also be noted at this point that there is no-universal accepted criterion for defining the exact limits of the transition zone between the near and far-fields; in general the use of only a 'near-field

model' has been adequate to determine whether required temperature standards are being met. However, in order to assess the total environmental impact the 'far-field' effects also require evaluation [44].

During the past several years various research programs in a number of countries have focused on the development of suitable mathematical models that can be used to predict the physical extent of heated effluents in all types of water bodies. For example, models for describing the physical characteristics of surface and submerged thermal discharges have been developed at the U.S. Atomic Energy Commission and Environmental Protection Agency laboratories, a number of universities and elsewhere. Extensive efforts have been carried out during the past years to prepare state-of-the-art reports on mathematical models, hydraulic models, and proto-type field data which have been collected to verify both analytical and physical models [19]. These reports describe the mechanics of model development and application and also provide a technical basis for assessing the ecological impact of thermal releases on the receiving water systems.

Mathematical or Numerical methods require dividing the area of interest into a large number of finite volumes or elements. Fluid properties such as velocity, density, pollutant concentration, and temperature vary from one element to the next but are assumed constant within the element. The boundaries in a numerical methods problem are usually simulated to represent the actual boundaries in the problem. Three dimensional solid boundaries require special representation of the elements in contact with the boundary. Usually some simplifying assumptions are made to make the problem easier to set up. Fluid boundaries are taken far enough away that the discharge of concern will not affect them. At these boundaries, fluid properties can either be fixed at a known value or made variable with time with some known variation.

The flow field must first be divided into the appropriate grid system. There are numerous programs available whose sole purpose is to generate grid systems for complicated problems. The differential equations of fluid motion are discretized and solved numerically for this system of elements with a computer. Various methods of discretization and solution are used. The most critical values are how the boundaries are simulated and the turbulence model used. One of the more popular model is the k-epsilon ($k-\in$) model. Computer run time for each case can vary from a few minutes to a few weeks depending on the type of problem and size of the computer used to make the computations. Output includes fluid properties at each element for a steady state problem or for each element at a finite number of time steps for a transient problem. These can be used to plot velocity, temperature, and concentration profiles in the flow field at any viewing plane.

The advantages of numerical methods are the ability to represent actual geometry and time dependency without having to go to the laboratory and set up a physical model. Disadvantages include the time required to set up and run a problem having a complicated boundary and the inability to accurately represent turbulence numerically. A Computational Fluid Dynamics (CFD) model approach is the most comprehensive and accurate model approach. It computes all flow characteristics at defined points in a grid, but the computational time is longer. Therefore, this model approach is less commonly used in the near field [2, 43].

Basic description of mathematical modelling

To study the circulation features in the study area at base line condition, Hydrodynamic (HD) Module was used. The various parameters considered in Hydrodynamic (HD) module are wind, tide, salinity, and temperature [50].

Thermal and Salinity Dispersion Study has been carried out using Advection-Dispersion (AD) module. The various parameters need to be considered in Advection-Dispersion (AD) module are circulation features from the Hydrodynamic (HD) Module (Water level, Currents) and the characteristics of the discharge outfall (Temperature, Salinity). The methodology has been given in Figure 9.

On the basis of a general consideration of diffusion phenomena of thermal discharge it is noted that, in order to predict the thermal diffusion of cooling water discharged into the sea, account should be taken of the hydraulic and thermodynamic behaviour of the released water. Therefore, three fundamental sets of equations are used for the analysis, namely equations of motion considering eddy viscosity; equations of continuity; and thermodynamic equations including heat budgets and heat exchange between these a surface and the atmosphere. Solving numerically these simultaneous equations under the boundary conditions concerning factors such as quantity, velocity and temperature of released cooling water, topography of the coast, location of the outlet, natural structure of temperature in the sea region, meteorological parameters (wind, solar insolation, air temperature, humidity, cloud cover, etc.), characteristics of turbulence in the sea, and maritime conditions (tidal, coastal and wind driven currents and waves) [50], one can obtain a detailed distribution both of velocity and temperature in the sea region in front of the outlet for each power station.

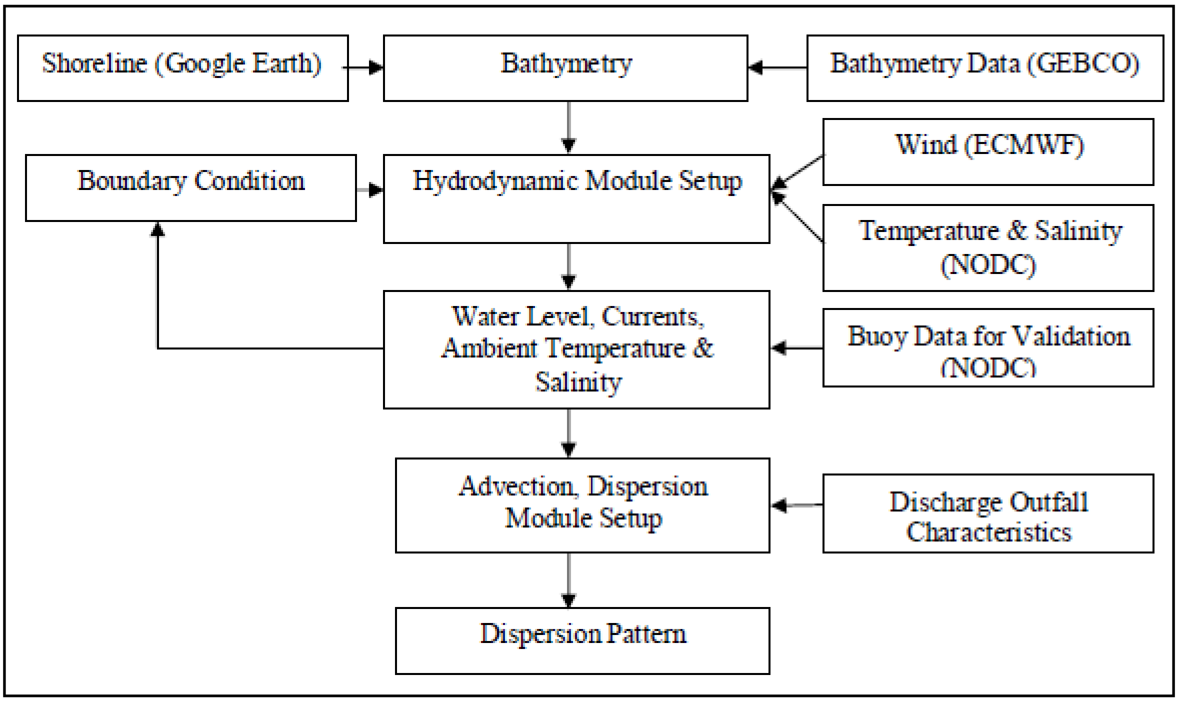


Fig. 9 Model methodology

Different types of Models

Since last two decades mathematical modelling has become a very efficient tool for studying coastal engineering problem. These models are essentially based on conservation of mass and momentum equation. Similar to scale model, there are tidal model as well as wave model. These models can be three dimensional (3D) or two dimensional (2D) depending upon governing equation. At present for coastal engineering problems mostly two dimensional models are used. The governing equations for toxicant like warm water are generally solved using different numerical methods such as finite difference method (FDM), finite element method (FEM), spectral method and finite volume method (FVM) as in the laterally integrated hydrodynamics and mass transport model. The independent variables are longitudinal distance, flow depth, and time. The water surface elevation and horizontal momentum are computed simultaneously in the model based on a numerical scheme, which allows the use of a reasonable time scale for field application over entire stratification cycles. Numerical models are used for simulation of hydrodynamics, dispersion of warm water and plume in waters. Mathematical models are generally developed for one-dimensional, two-dimensional and three-dimensional free surface flows. There are number commercial and scientific software available for this like MIKE, Flow3D, Telemac, OpenFOAM, SU2, Delft3D, EFDC etc.

Herein we will take MIKE software for discussion on this. Numerical model software, MIKE21AD simulates dispersion of warm water in coastal environment. The modelling software is developed by Danish Hydraulics Institute (DHI) [35], Water and Environment, Denmark and is one of the widely used commercial software It is based on the non-linear vertically integrated 2-D equation of conservation of mass, which takes into account advection and dispersion. Mathematical modelling software, MIKE 3 is also used for simulations of coastal hydrodynamics and dispersion of warm water in the study area. The modelling software is also developed by DHI [36]. In order to assess the circulation pattern of warm water, hydrodynamic modelling (MIKE 3 HD) and to determine movement of thermal plume, Advection-Dispersion modelling (MIKE 3 AD) is used. Both the modules are included in MIKE 3 model.

4.3 Brief description of model

4.3.1 Physical model

Due to the complexity of the phenomenon of mixing of warm water discharge into the ambient receiving body of water, physical modelling of the process poses several problems. Ideally, it is necessary to model three regions – the near field, the region of development and the far field. All these regions cannot be combined in one physical model because the vertical scale of the model needs to be large enough to allow room for differentiating stratified layers, to prevent surface tension forces from controlling the dynamics of flow and to ensure turbulent Reynold's number. In case of geometrically similar models, the surface area would be so large as to make the model unwieldy and the radiation losses would be much higher than the actual. Moreover, the modelling considerations of a physical model involving simulations especially of the tidal phenomenon, needed to be vertically exaggerated.

Similitude requirements for heated discharges

A dimensionless formulation of the heated discharge shows that the induced temperature T is a function of several parameters as given below:

$$\frac{T - T_a}{T_o - T_a} = f\left(F_o, A, \frac{K}{\rho C U_o}, \frac{V}{U_o}, \frac{U_o h_o}{\upsilon}\right) \qquad (19)$$

T_o = Temperature of the heated discharge (^oC)

T_a = Ambient water temperature (^oC)

F_o = Densimetric Froude Number = $\dfrac{U_o}{\sqrt{\left(\dfrac{\Delta\rho}{\rho_a}\right)(gh_o)}}$

A = Discharge channel aspect ratio = $\dfrac{h_o}{b_o}$

K = Surface heat loss parameter

$\dfrac{V}{U_o}$ = Cross flow parameter

U_o = Discharge Velocity (m/sec)

$\Delta\rho$ = Density difference between the heated discharge and the ambient water (kg/m^3)

G = Gravitational acceleration (m/sec^2)

ρ_a = Density of ambient water at temp. T_a (kg/m^3)

υ = Kinematic Viscosity of heated water (m^2/sec)

h_o = Depth of the heated discharge (m)

Froude Criteria

The jet induced entrainment and temperature distribution in the immediate vicinity of the outlet are found to be the function of the densimetric Froude Number and geometric parameters such as aspect ratio and bottom slope. In a vertically distorted Froude model, the ratio of Froude Number in the model to that in the prototype has to be 1. In order to satisfy the Froude Similitude, it follows that in a vertically distorted model, the velocity scale has to be the root of the length scale $\left(V_r = \sqrt{Lz_r}\right)$.

The Froude Number similarity implies that

$$\left(\frac{U_o}{\sqrt{gh_o}}\right)_{\mathrm{Proto}} = \left(\frac{U_o}{\sqrt{gh_o}}\right)_{\mathrm{model}} \qquad (20)$$

Similarly the densimetric Froude Number similarity criteria implies that

$$\left(\frac{U_o}{\sqrt{\left(\frac{\Delta\rho}{\rho_a}\right)gh_o}}\right)_{\mathrm{Proto}} = \left(\frac{U_o}{\sqrt{\left(\frac{\Delta\rho}{\rho_a}\right)gh_o}}\right)_{\mathrm{model}} \qquad (21)$$

Hence it follows that

$$\left(\frac{\Delta\rho}{\rho_a}\right)_{\mathrm{Proto}} = \left(\frac{\Delta\rho}{\rho_a}\right)_{\mathrm{model}} \qquad (22)$$

It will thus be seen that in order to satisfy the similitude requirements under both the Froude Number and densimetric Froude Number, the ratio of $(\Delta\rho/\rho_a)=1$. With proper selection of scales, one can have the temperature rise above the ambient in the model should be the same as that in the prototype. It facilitates interpretation of model results with ease as the rise in water temperature above the ambient observed on the model gives directly the expected rise in ambient water temperature in prototype.

Reynold's Number

To ensure turbulent flow, the Reynold's Number in the model should be greater than about 2500 while designing the tidal models, and scales are required to be chosen accordingly.

Similarity requirement for surface heat loss

The modelling law for surface heat loss can be obtained based on the one-dimensional equation for heat transport. The following equation applies for steady state flow.

$$\left(\frac{T_X - T_E}{T_O - T_E}\right) = Exp.\left(\frac{BxK}{Q\rho C}\right) \qquad (23)$$

Where

T_X = Water Temperature as a function of space and time(^oC)

T_E	=	Equilibrium Temperature(^oC)
T_O	=	Temperature of heated discharge at the point of out let(^oC)
K	=	Heat transfer Coefficient (K Cal/m^2/^{o}C)
Q	=	Flow rate of heated discharge (m^3/sec)
B	=	Width of Channel (m)
ρ	=	Density of Water (kg/m^3)
C	=	Specific Heat of water (Kcal/kg ^{o}C)
x	=	Horizontal Distance (m)

For similarity, assuming the same fluid in the model and in the prototype, it is obtained that

$$\left(\frac{Lx^2{}_r}{Q_r}\right)(K_r)=1 \tag{24}$$

Where Lx_r = Ratio of the horizontal length scale

K_r = Ratio of heat transfer co-efficient in the model and prototype

The Froude criterion still applies and as such,

$$Q_r = L^{3/2}z_r \; Lx_r \tag{25}$$

Where $L\,z_r$ = Ratio of vertical length scale

Thus, $$\frac{L\,z_r}{L\,x_r}=\frac{K_r}{(L\,z_r)^{1/2}} \tag{26}$$

Since wind effect on model is insignificant, it has been the practice to have $K_r \approx 0.5$, as generally recommended in literature

4.3.2 2D & 3D Mathematical model

4.3.2.1 MIKE 21 model

MIKE 21 HD Module

2-Dimensional hydrodynamic module, MIKE-21 HD [35] is used to simulate the flow field in the thermal or nuclear power project site under different oceanographic and meteorological conditions.

The conservation of mass and momentum integrated over the vertical are used in the hydrodynamic model to describe the flow and water level variations

$$\frac{\partial H}{\partial t}+\frac{\partial p}{\partial x}+\frac{\partial q}{\partial y}=0 \tag{27}$$

$$\frac{\partial P}{\partial t}+\frac{\partial}{\partial x}\left(\frac{p^2}{h}\right)+\frac{\partial}{\partial y}\left(\frac{pq}{h}\right)+gh\frac{\partial H}{\partial x}+\frac{gp\sqrt{p^2+q^2}}{c^2h^2}$$
$$-\frac{1}{\rho w}\left[\frac{\partial}{\partial x}(h\tau xx)+\frac{\partial}{\partial y}(h\tau xy)\right]-\Omega q-fVV_x+\frac{h}{\rho w}\frac{\partial}{\partial x}(P_a)=0 \tag{28}$$

$$\frac{\partial q}{\partial t}+\frac{\partial}{\partial y}\left(\frac{q^2}{h}\right)+\frac{\partial}{\partial x}\left(\frac{pq}{h}\right)+gh\frac{\partial H}{\partial y}+\frac{gq\sqrt{p^2+q^2}}{c^2h^2}-\frac{1}{\rho w}\left[\frac{\partial}{\partial y}(h\tau_{yy})+\frac{\partial}{\partial x}(h\tau_{xy})\right]-\Omega p-fvv_y$$
$$+\frac{h}{\rho w}\frac{\partial}{\partial y}(P_a)=0 \tag{29}$$

The following symbols are used in the equations:

h(x, y, t)	:	water depth(m)
H(x, y, t)	:	surface elevation (m)
p,q(x,y,t)	:	flux densities in x and y directions [m^3/(s.m)]
c(x,y)	:	Chezy resistance ($m^{1/2}$/s)
g	:	acceleration due to gravity (m/s^2)
f(v)	:	wind friction factor
v, v_x, v_y(x,y,t)	:	wind speed and components in x and y direction (m/s)
Ω(x,y)	:	Coriolis parameter (s^{-1})
P_a	:	atmospheric pressure (kg/ms^2)
ρw	:	density of water (kg/m^3)
x, y	:	space coordinates (m)
t	:	time(s)
τ_{yy}, τ_{xy}, τ_{xx}	:	components of effective shear stress (N/m^2)

In order to simulate dispersion of temperature, it is necessary to initially compute the hydrodynamics of water body in terms of velocity and water level fluctuations. The appropriate governing equations for hydrodynamics in tidal areas are given by the shallow water wave equations. These two dimensional shallow water equations are derived from the Navier Stokes equations of motion with the following simplified assumptions:

The flow is incompressible

The flow is well mixed

Vertical accelerations are negligible

Bed stress can be modeled using a quadratic friction law.

These unsteady governing equations are solved using non-iterative Alternating Direction Implicit (ADI) finite difference technique. ADI implies that in one time step, there are two half time steps. In one half time step, the scheme is implicit in the east-west direction and explicit in the north-south direction. In the next half time step, computations are implicit in the north-south direction and explicit in the east-west direction. ADI scheme is computationally efficient and widely used in solving shallow water equations.

MIKE 21 AD Module

The advection dispersion equation is solved for dissolved or suspended substances in two dimensions. It is in fact the mass-conservation equation. Discharge quantities and compound concentrations at source and sink points are included together with a decay rate.

$$\frac{\partial}{\partial t}(hc)+\frac{\partial}{\partial x}(uhc)+\frac{\partial}{\partial y}(vhc)=\frac{\partial}{\partial x}\left(hD_x\frac{\partial c}{\partial x}\right)+\frac{\partial}{\partial y}\left(hD_y\frac{\partial c}{\partial y}\right)-Fhc+S \tag{30}$$

The following symbols are used in the equation:

c	:	compound concentration (arbitrary units)
u,v	:	horizontal velocity components in x, y directions (m/s)
h	:	water depth (m)
D_x,D_y	:	dispersion coefficients in the x,y directions (m^2/s)
F	:	linear decay coefficient (sec^{-1})
S	:	$Q_s(c_s-c)$
Q_s	:	source/sink discharge ($m^2/s/m^2$)
c_s	:	concentration of compound in the source/sink discharge (unit)

Information on u, v and h at each time step is provided by the hydrodynamic module.

The most common use for a heat dissipating substance is that of modelling power plant cooling water recirculation. Here, the advected and dispersed component is heat (or rather temperature). The heat dissipation follows a prescribed function where the decay factor F may take the following form:

$$F = 0.2388/(\rho.Cp.h).[(4.6-0.09(T_{ref}+T_e)+4.06w).\exp(0.033(T_{ref}+T_e))] \quad (31)$$

Which is suitable for heat dissipation to the atmosphere.

Where

w	:	wind speed (m/s)
T_{ref}	:	the reference temperature (^oC)
T_e	:	excess temperature (^oC)
ρ	:	density of water (kg/m^3)
Cp	:	specific heat (cal/kg ^oC)

It uses a third-order finite difference scheme QUICKEST which in many ways has very fine qualities. It avoids the wiggle instability problem associated with central differencing of the advection terms, and at the same time it eliminates the numerical damping often experienced with first order up winding methods. The scheme itself is a Lax-Wendroff like scheme in the sense that it cancels out any truncation error terms due to time differencing up to a certain order by using the basic equation itself. In the case of quickest, truncation error terms up to third order are cancelled, that is for both space and time derivatives.

4.3.2.2 MIKE 3 model

MIKE 3 model is non-hydrostatic and applies an Artificial Compressibility Method (ACM). The mathematical foundation in MIKE 3 [36] is the mass conservation equation, the Reynolds-averaged Navier-Stokes equations in three dimensions including the effects of turbulence and variable density together with conservation equations for salinity and temperature. It simulates unsteady three dimensional flows taking into account density variations, bathymetry and external forcing such as meteorology, tidal elevation, currents and other hydrographic conditions. In three dimensional hydrodynamic model for flow of Newtonian fluids, following elements are required:

- Mass conservation
- Momentum conservation
- Conservation of salinity and heat
- Equation of state relating local density to salinity, temperature and pressure

Thus, the governing equations consist of seven equations with seven unknowns.

$$\frac{1}{\rho c_s^2}\frac{\partial P}{\partial t}+\frac{\partial u_i}{\partial x_j}=SS \tag{32}$$

$$\frac{\partial u_i}{\partial t}+\frac{\partial\left(u_i u_j\right)}{\partial x_j}+2\Omega_{ij}u_j=-\frac{1}{\rho}\frac{\partial P}{\partial x_i}+g_i+\frac{\partial}{\partial x_j}\left[\nu_T\left(\frac{\partial u_i}{\partial x_i}+\frac{\partial u_j}{\partial x_j}\right)-\frac{2}{3}\delta_{ij}k\right]+u_i SS \tag{33}$$

$$\frac{\partial S}{\partial t}+\frac{\partial}{\partial x_j}\left(Su_j\right)=\frac{\partial}{\partial x_j}\left(D_S\frac{\partial S}{\partial x_j}\right)+SS \tag{34}$$

$$\frac{\partial T}{\partial t}+\frac{\partial}{\partial x_j}\left(Tu_j\right)=\frac{\partial}{\partial x_j}\left(D_T\frac{\partial T}{\partial x_j}\right)+SS \tag{35}$$

where ρ is the local density of fluid, c_s the speed of sound in seawater, u_i, the velocity in the x_i – direction, Ω_{ij} the Coriolis tensor, P the fluid pressure, g_i the gravitational vector, νT the turbulent eddy viscosity, δ Kronecker's delta, k the turbulent kinetic energy, S and T the salinity and temperature, D_s and D_t the associated dispersion coefficients and t denotes time, SS refers to respective source-sink terms and thus differs from equation to equation. The salinity, temperature and pressure are related to density through the United Nations Educational, Scientific and Cultural Organization (UNESCO) definitions.

Equations (1) and (2) are referred to as hydrodynamic equations which are solved in Hydrodynamic Module (HD) whereas equations (3) and (4) are referred to as advection-dispersion equations which are solved in Advection-Dispersion module (AD).

Hydrodynamic Module (HD) of MIKE 3 makes use of the so-called Alternating Direction Implicit (ADI) technique to integrate the equations for mass and momentum conservation in space-time domain. The equation matrices, which result for each direction and each individual grid line, are resolved by a Double Sweep (DS) algorithm. The difference terms are expressed on a staggered grid in x, y and z-space.

In MIKE 3, transport of scalar quantities; salinity and temperature, are handled in Advection-Dispersion module. General advection-dispersion equation is expressed as

$$\frac{\partial C}{\partial t}+\frac{\partial u_i C}{\partial x_i}=\frac{\partial}{\partial t}\left(D_i\frac{\partial C}{\partial x_i}\right)+SS \tag{36}$$

in which C is the scalar concentration variable, Di the dispersion coefficient and SS represents source-sink term.

MIKE 3 offers four different advection-dispersion schemes:

(i) The fully 3D QUICKEST-SHARP scheme which is especially suitable for simulations with steep gradients

(ii) The ULTIMATE-QUICKEST scheme with operator splitting and intermediate surface elevations calculated on basis of locally 1D continuity equations. The ULTIMATE-QUICKEST scheme is an alternative to the QUICKEST-SHARP scheme and it is designed to reduce computation time when more than one component has been selected

(iii) The simple UPWIND scheme which is similar to the ULTIMATE-QUICKEST scheme except the up-winding is applied all over

(iv) The fully 3D UPWIND scheme which is similar to the QUICKEST-SHARP scheme except the up-winding is applied all over

The first scheme being more advanced is used generally.

The information of u_i at each time step is provided by hydrodynamic module and thus assumed constant throughout the time integration of the advection-dispersion equation. The advection-dispersion equation is solved at each time step following the time integration of the hydrodynamic equations. The nested version of MIKE 3 HD (MIKE 3 NHD) has a built-in refinement-of-scale facility, which gives a possibility of increasing the resolution in areas of special interest. As with the standard MIKE 3 HD, the mathematical foundation in MIKE 3 NHD is the mass conservation equation, the Reynolds-averaged Navier-Stokes equations including the effect of turbulence and variable density together with the conservation equations for salinity and temperature.

The equations are spatially discretised on a rectangular staggered grid, the so-called Arakawa C-grid. Scalar quantities such as pressure, salinity and temperature are defined on the grid nodes whereas velocity components are defined halfway between adjacent grid nodes in respective directions. The method applied to ensure dynamic nesting in MIKE 3 NHD, i.e. the two-way dynamic exchange of mass and momentum between the modelling grids of different resolution, is a relatively simple extension of solution method used in the standard hydrodynamic module.

4.4 How are the results interpreted?

Major model input includes:

(a) Domain and time parameters such as bathymetry with computational mesh, total simulation length and time step;

(b) Calibration parameters such as bed resistance, dispersion factor and wind friction factor;

(c) Initial conditions in terms of water surface elevation, velocity components and temperature /salinity concentration;

(d) Boundary conditions in terms of tidal water level, flux, creek discharge and temperature /salinity concentration;

(e) Other driving forces like wind speed and direction, isolated source /sink discharge, cloud cover, air temperature and humidity.

Further, locations of intake-outfall structures, rise in excess temperature/ salinity at outfall are also required to be provided as input to the model. Model output includes value of the basic variables consisting of water depth and surface elevation, flux densities and velocities in main directions, density, temperature and salinity at each time step at every mesh element.

4.5 Importance of exact implementation of design at site

Accuracy is a parameter that has to be kept in practice from the initial stage of planning to the end of the project. In particular, nuclear power projects are quite sensitive projects as they deal with the radioactive materials. Also, there are many costs involved at each and every stage. Hence, even the smallest of a mistake in the execution of the design at site may lead to a heavy financial and maybe even loss of life. Hence, utmost care needs to be taken while executing the thermal/nuclear power plant establishments owing to environmental considerations.

Chapter 5

CASE STUDIES

5.1 Introduction

Several physical model and 2-D & 3-D mathematical model studies undertaken and completed successfully at CWPRS for locating intakes and outfalls for various thermal and nuclear power projects have been enumerated herein. Major studies have been carried out are for thermal and nuclear plants viz., North Chennai (NCTPS), Ennore (ETPS), Kalpakkam , Kudankulam (KKNPP), Jaitapur (JNPP), Kovvada, Tarapur etc. Some of them are discussed herein as case studies.

5.2 Thermal power plants

5.2.1 L*ocating intake and outfall structures of cooling water system for North Chennai Thermal Power Station (NCTPS), Tamil Nadu using physical modelling techniques*

Background

Tamil Nadu Electricity Board (TNEB) has established two units of thermal power stations at Ennore which is about 15 km North of Chennai. One unit is Ennore Thermal Power Station (ETPS) of generating capacity 450 MW located at about 4.5 km South of the Ennore mouth and the other unit is North Chennai Thermal Power Station (NCTPS) of generating capacity 630 MW located at about 1.5 km North of the mouth. Location map is shown in Figure 10. Both were established in 1994. The ETPS requires 23 cumecs of water for cooling condensers drawn from the Korttalaiyar River, a branch of Ennore creek system. The hot water is discharged back to sea through a pipe running on jetty. Similarly, NCTPS requires 27.5 cumecs of cooling water drawn through an intake channel from the Ennore creek. Hot water from condensers is let out to sea through a surface discharge system located at about 2 km North of the Ennore mouth.

Proposal

Due to various difficulties prevailing at site, it was observed that the Ennore mouth was getting choked very frequently affecting the water supply severely. In view of this, physical model studies were conducted at CWPRS for suggesting remedial measures to improve the conditions.

Studies

Following five alternative proposals were studied on physical model as depicted in Figure 11;

Alternative 1: Intake in Ennore Port basin, outfall on the north of northern breakwaters.

Alternative 2: Intake in Ennore Port basin, outfall at the mouth of the creek.

Alternative 3: Intake in Ennore Port basin, outfall located at Pulicat backwaters.

Alternative 4: Intake at existing location, outfall on the north of northern Breakwaters.

Alternative 5: Intake at existing location, outfall at the mouth of the creek.

The Model

The physical model is mainly for simulation of “far field reach of the thermal discharges”. The model scales are selected to 1:360 horizontal and 1:80 vertical. With these scales, the other relevant scales worked out as follows:

Equality of Froude’s number is a must for tidal/quasi steady flow model. The scale relationships are derived based on these criteria.

$$\text{Froude's Number } (F_e) = \frac{V}{\sqrt{gL_{zr}}} \tag{37}$$

$$\frac{V_m}{\sqrt{g\,L_{zm}}} = \frac{V_p}{\sqrt{g\,L_{zp}}} \tag{38}$$

$$\frac{V_m}{V_p} = \sqrt{\frac{(L_z)_m}{(L_z)_p}} \tag{39}$$

$$V_r = \sqrt{L_{zr}} \tag{40}$$

Other scale laws in terms of length and depth scales are as follows

$$\text{Time scale } T_r = \frac{L_r}{V_r} = \frac{L_{xr}}{\sqrt{L_{zr}}} \tag{41}$$

$$\text{Discharge scale } Q_r = A_r V_r = L_{xr}\, L_{zr} \sqrt{L_{zr}} \tag{42}$$

$$Q_r = L_{xr}\,(L_{zr})^{\frac{3}{2}} \tag{43}$$

Where V_r = Velocity scale ratio

Lx_r = Ratio of the horizontal length scale

$L\,z_r$ = Ratio of vertical length scale

Using these relations following scales were evolved

Velocity scale 1:8.94

Time scale 1:40.27

Discharge scale 1:2, 57, 595

The total requirement of electrical heat input for reproducing hot water discharge in the model was 6 KW, for obtaining 5°C rise in water temperature above the inlet temperature. The tidal period in model was 18 min. 25 s equivalent to 12 hours 25 min. in prototype.

Test Conditions

The Oceanographic parameters considered for the physical model studies for hot water recirculation were (i) the tidal levels during spring and neap tides and, (ii) the tidal currents.

Outcome

Based on the results of the physical model studies conducted for different alternatives [3], Alternative 2 was found suitable. The results of Alternative 2 confirmed that the temperature of the hot water at the outlet was 5°C above the ambient. At low water level, the warm water flows towards the ETPS and the drop in temperature was observed to be from 5 °C near the outfall, to 0.5 °C near Railway Bridge over Korttalaiyar backwaters over a distance of about 2.0 km. The temperature drop near the existing intake was observed 1 °C during the low water level, and 0.5 °C during high water level as depicted in Figure 12.

Based on the extensive studies on the model, it was concluded that:

1. The proposal of locating the intake in the Ennore port basin and outfall at the mouth of the creek would be the most suitable solution i.e. Alternative 2.

2. The above proposal restores the imbalance in the tidal influx and efflux. In fact, the tidal efflux would be marginally higher than the tidal influx. As such, in the long run, the quantity of dredging required to keep Ennore mouth open would go on reducing and the mouth would attain a stable form.

3. Due to location of warm water discharge in the creek, the water level in the creek would increase and the ETPS would also get an assured supply of cooling water.

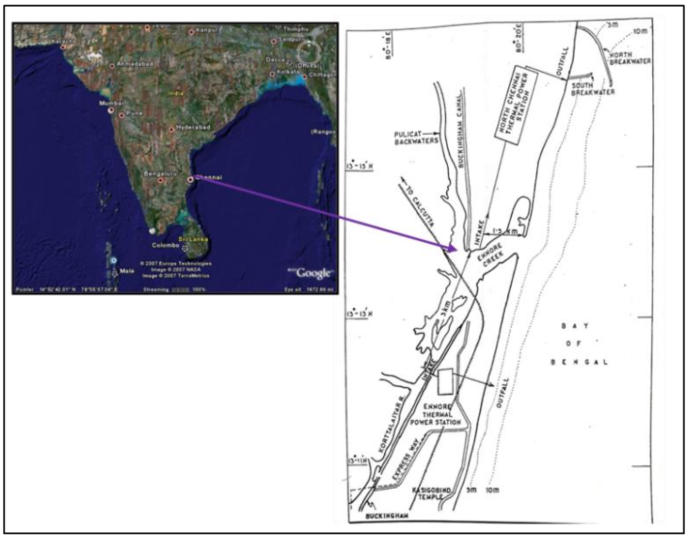


Fig. 10 Location plan of NCTPS and ETPS

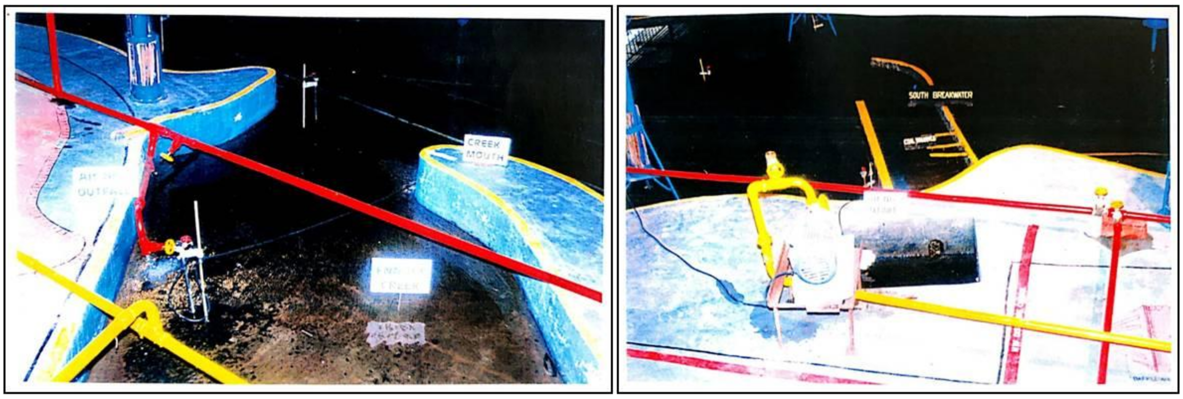


Fig. 11 Layout of cooling water intake and warm water outfall at model

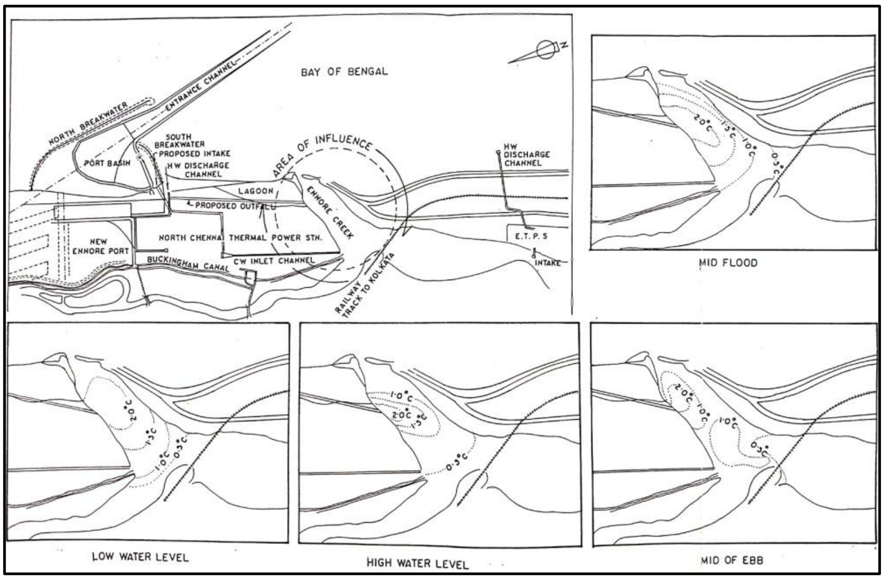


Fig. 12 Isotherms of recommended Alternative 2 obtained in the model experiments

5.2.2 *Determination of location of intake and outfall of proposed thermal power plant at Salav, Maharashtra using mathematical modelling techniques*

Background

M/s Welspun Max Steel Ltd (WMSL) proposed a steel plant located at Salav in Raigad district which is at about 130 km south of Mumbai, Maharashtra. The plant is to be located on the left bank of Kundlika (near Revadanda) river which is a creek meeting to sea on the west at about 4-5 km.

Proposal

WMSL is also planning to develop a 330 MW coal based captive power plant for operating their steel plant at Salav. The cooling water required for the power plant is proposed to be withdrawn from creek of Kundlika river and the warm water would be discharged back to the creek at a suitable location in the creek. The tentative locations of the intake and outfall are shown in Figure 13. The circulation water (CW) system proposed for the plant is closed circuit re-circulating type system with induced Draft Cooling Tower. The makeup water would be drawn from creek and warm brine water would be let out to the creek. The inflow from intake would be 2409 m^3/hr and the outflow from the outfall would be 1875 m^3/hr. At the outfall, the rise in temperature would be 5 ^{0} C [7].

Studies

The main scope of the mathematical model studies was to simulate dispersion of warm water discharge from the thermal power plant using two dimensional mathematical models MIKE21 HD and AD and to confirm suitability of given locations of intake and outfall with minimum recirculation in the intake under various discharge conditions.

Outcome

Based on the 2D mathematical model studies it was found that the thermal plume grew with time towards east and west directions depending upon the tidal conditions. The tide and tidal current played a significant role on the spreading of the thermal plume and its temperature rise. The intake and outfall temperature values varied with tidal range and tidal current speed. It was found from the results obtained from mathematical model simulations that the withdrawal and discharge back to creek which were very small (only 0.67m^3/s and 0.41 m^3/s respectively) in comparison to the tidal flow in the Creek. From the thermal model dispersion studies it was found that the maximum rise in excess water temperature at the intake as well as outfall is less than 0.25^0 C as shown in Figure 14. With the given location of Intake II and the outfall, the recirculation of excess temperature in the intake is minimum and within the permissible limit. Further, it was also observed that the outfall discharge with rise in excess temperature do not get accumulated in the creek at any stages of power plant operation [7].

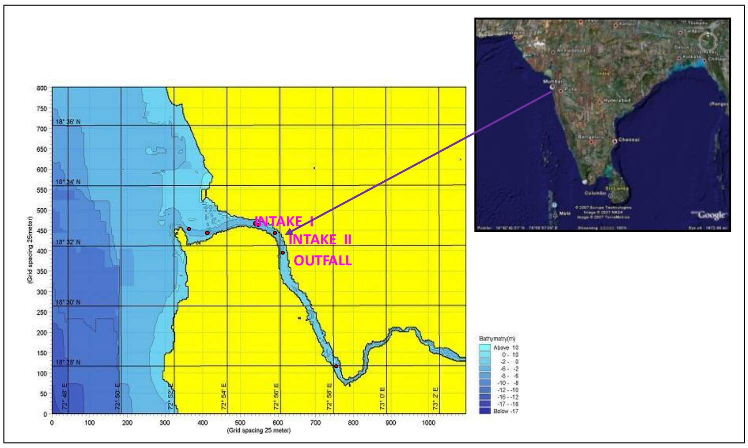


Fig. 13 Location map and location of proposed intakes & outfall at thermal power plant at Salav

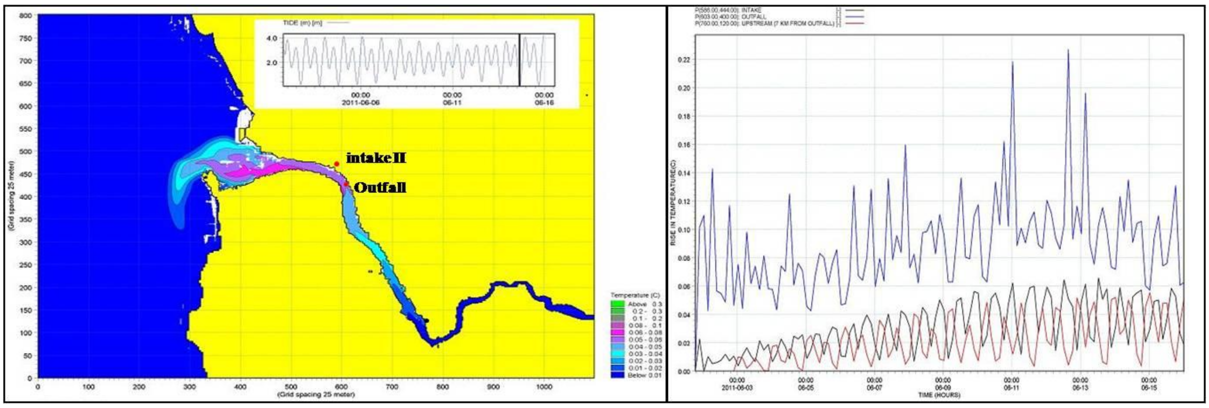


Fig. 14 Distribution and time history of rise in excess temperature at intake and outfall

5.3 Nuclear power plants

5.3.1 Two dimensional mathematical model studies

5.3.1.1 *Determination of locations of intake/outfall for PFBR at Kalpakkam, Tamil Nadu*

Background

Kalpakkam is located on the east coast, about 70 km south of Chennai in Tamil Nadu State. Mahabalipuram, the famous tourist place is about 5 km north of Kalpakkam. Location map is shown in Figure 15. The Madras Atomic Power Station (MAPS) having two units of 220

MW was established in 1983-85 by Nuclear Power Corporation of India Ltd (NPCIL). MAPS is operating on once-through cooling water system for which the cooling water required for the power station (i.e. about 35 m^3/s) is drawn from the sea through an intake well which is located at 360 m from the shoreline and connected to the forebay of the pump house by a submarine tunnel. An approach jetty is constructed for carrying out maintenance of the intake well. The warm water from the condensers with rise in temperature of 10^0 C is discharged back to sea through a triangular shaped outfall structure located at the root of the approach jetty.

Proposal

Department of Atomic Energy has proposed to establish a Fast Breeder Reactor Project (PFBR) of 500 MW capacity at a location 500 m south of MAPS. the location of intake structure of PFBR which is to be located at about 680 m towards South from the MAPS intake structure. The distances of MAPS intake and PFBR intake from the coastline are 360 m and 460 m respectively as shown in Figure 16. The MAPS outfall point is about 30 m near coastline. The cooling water requirement for this proposal was considered to be 22 m^3/s and the warm water from the condensers was assumed to be with rise in temperature of 10 ^{0}C. In view of the conditions of the intake/outfall system described above, a suitable location of outfall of PFBR was to be determined with minimum warm water recirculation in the intakes of MAPS and PFBR.

Studies

(a) Field data collection and analysis for Condenser Cooling Water System (CCWS) of PFBR [4].

(b) Mathematical model studies for location of intake/outfall for 500 MWe Prototype Fast Breeder Reactor at Kalpakkam, Tamil Nadu (Revised) [5].

(c) Supplementary mathematical model studies for littoral drift and thermal recirculation [6].

Outcome

The outflows from PFBR and MAPS to be let out through a channel parallel to coastline formed by constructing a guided bund. The channel would consist of a guided bund of 680 m length from the PFBR outfall to MAPS outfall and would extend further towards north by

500 m. It is seen that the littoral transport occurs up to 300 m from shoreline and also the likely shoreline advancement at the channel outlet would be of the order of 30 m. In view of this it would be desirable to locate the PFBR intake beyond about 350 m from shore. The approach jetty for carrying out maintenance of the intake well need to be constructed on open piles which would offer minimum obstruction to coastal currents and the littoral drift prevailing in the region. Thermal recirculation study indicated that combined outflow channel with guided bund of length 680 m from PFBR outfall to MAPS outfall and with minimum length of 500 m beyond MAPS outfall would be required for minimising recirculation of warm water in the intakes. This minimum channel length of 500 m is also required to keep the channel mouth sufficiently away from the MAPS intake and to avoid possible effect of coastline advancement due to the obstruction of the outflow to the littoral drift. Layout of MAPS and PFBR as per the CWPRS design is depicted in Figure 17. Typical temperature plume at surface for different current scenario due to warm water discharge from PFBR is shown in Figure 18.

The average width of the channel from PFBR outfall to MAPS outfall (680 m), of the channel beyond MAPS outfall (500 m) and of the channel mouth are determined to 15 m, 30 m, and 45 m respectively to ensure smooth flow conditions and to achieve velocities approximately in the range 1-2 m/s for avoiding scouring/deposition in the channel. Depth in the channel would be of the order of 1.0-1.6 m and the maximum water level would be less than +7.9 m. The average velocity in the two channels would be of the order of 1.2 m/s and 1.6 m/s when both MAPS and PFBR are operating [4, 5, 6].

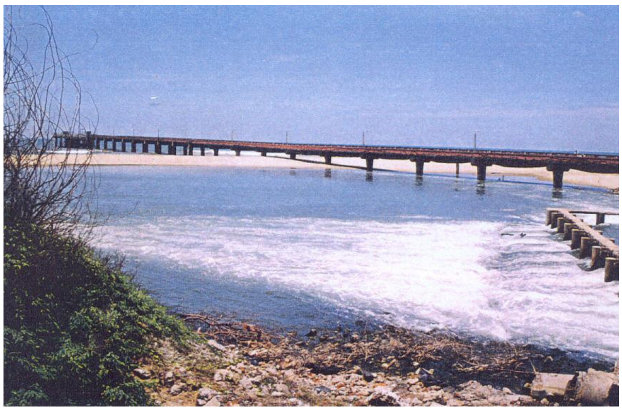


Fig. 15 Location Map of Kalpakkam

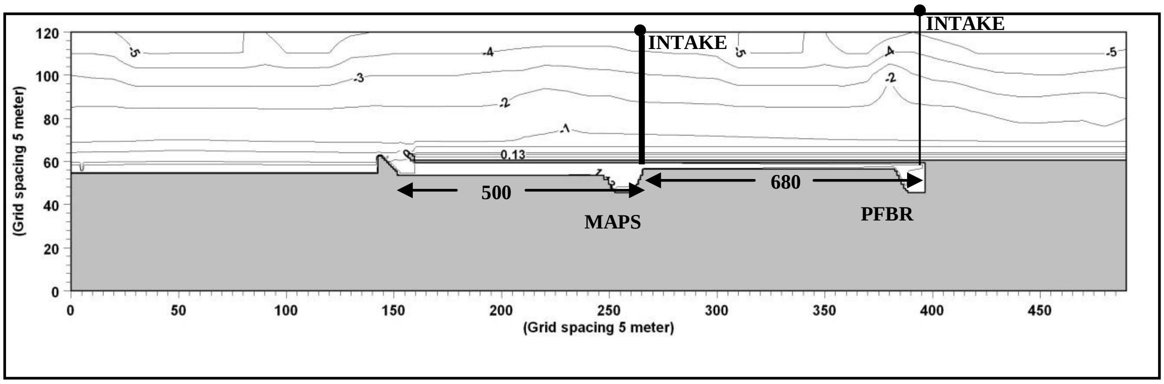
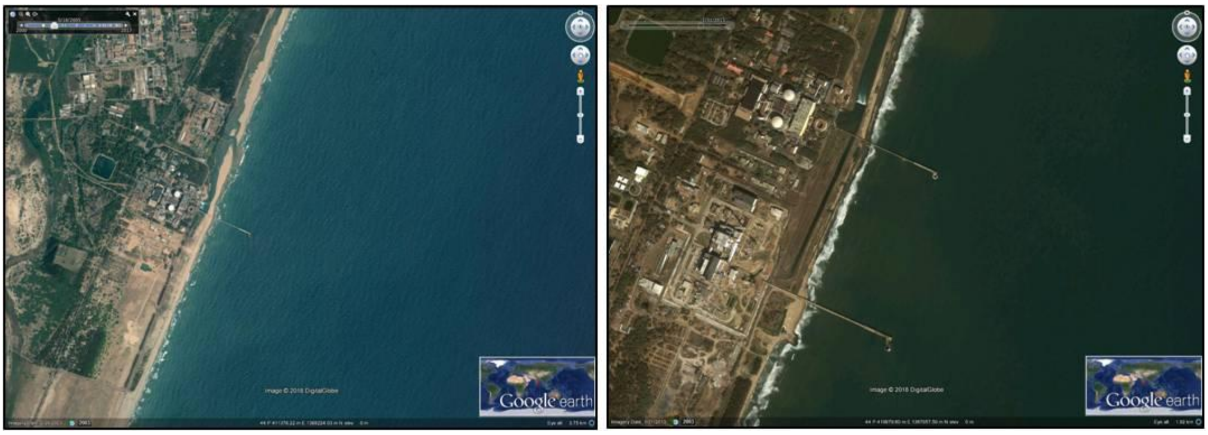


Fig. 16 Intake outfall system suggested based on model studies (2006)



(a) MAPS unit (2005) (b) MAPS and PFBR (2011)

Fig. 17 Locations of MAPS and PFBR as per CWPRS design

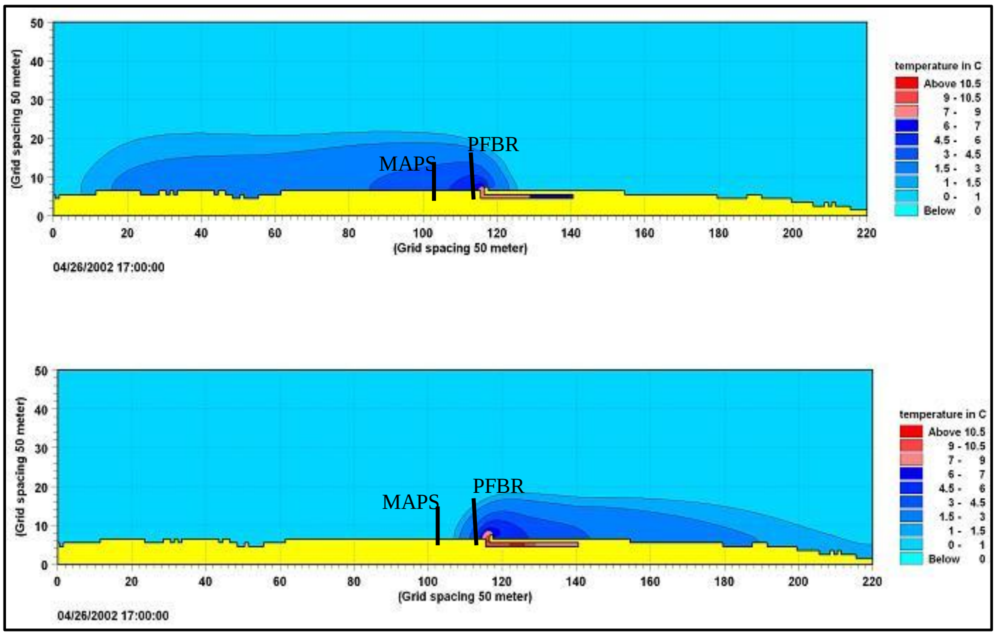


Fig. 18 Predicted Temperature plume at Surface

5.3.1.2 *Dispersion of warm water discharge from proposed nuclear power plant at Kovvada, Andhra Pradesh*

Kovvada nuclear power plant is one of the major nuclear power plants proposed to be established by Nuclear Power Corporation of India Limited (NPCIL), Mumbai at Kovvada situated at about 75 km North of Vishakhapatnam port in Ranasthalam block of coastal district Srikakulam, Andhra Pradesh. The proposed power plant is situated on the east coast of India. The power plant will have a generating capacity of 6 X 1000 MW comprising light water reactors.

Model bathymetry used in the simulations and predictions is depicted in Figure 19. The mathematical model studies for dispersion of warm water discharge from the proposed nuclear power plant at Kovvada for NPCIL, Mumbai were carried out considering two different Layouts of intakes /outfalls taking into account conditions in three different phases. A layout was provided by the Project Authority in which it was proposed that intakes to be located at shore in an intake channel and outfalls to be located in pairs in staggered manner in deep sea at 2.5 km, 3.0 km and 3.5 km from shoreline. In order to get the desirable /permissible temperature circulation at intake locations numerical experiments were carried out for the locations of intakes and outfalls in the given layout [9].

Hydrodynamic and thermal models simulation carried out for a period of 30 days continuously for the layout plan proposed by the Project Authority under different phases based on tidal current direction predominant at the proposed site during a year. During phase-I and phase-III tidal currents were predominant in south-west and north-east directions respectively whereas during phase-II the current movement was in both the directions. Based on simulation results it was seen that there was negligibly small rise in temperature above the ambient at intakes locations in the channel during phase-I and phase-III whereas in phase-II, considered to be the critical, small rise in intake temperature i.e. upto 2.5 ^{0}C was observed during the simulation.

The main conclusion of the mathematical model studies [9] is that the proposed layout of intakes, outfalls and inclined intake channel with locations of outfalls in staggered manner is suitable as it does satisfy the criteria of permissible limit of temperature circulation at intakes. However, during the transition phase of currents, which represents transitional phase of tidal currents at the site, the rise in temperature at intakes is about 2.5 ^{0}C above the ambient but as the period of occurrence of this phase is relatively less, such a small rise in intake temperature occurring for a short duration may be permissible with the existing location of the power plant. Variation of excess temperature and extent of thermal plume in critical condition is shown in Figure 20.

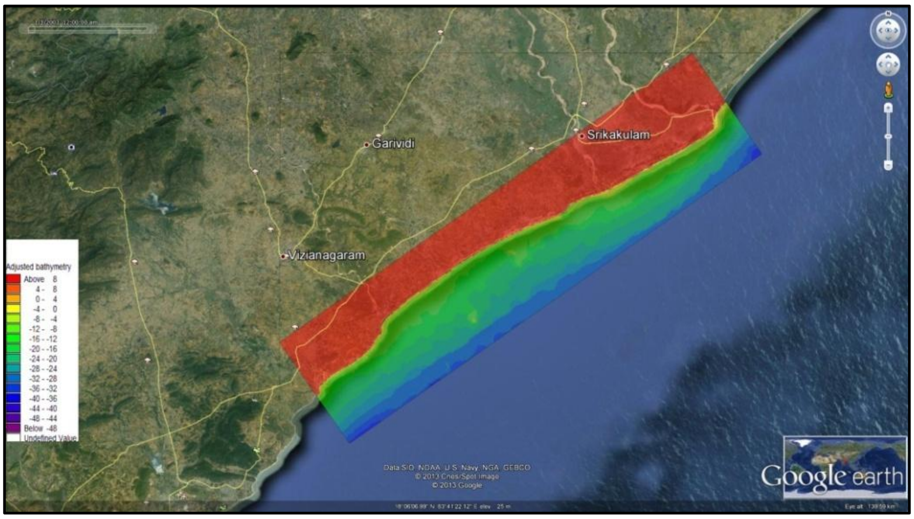
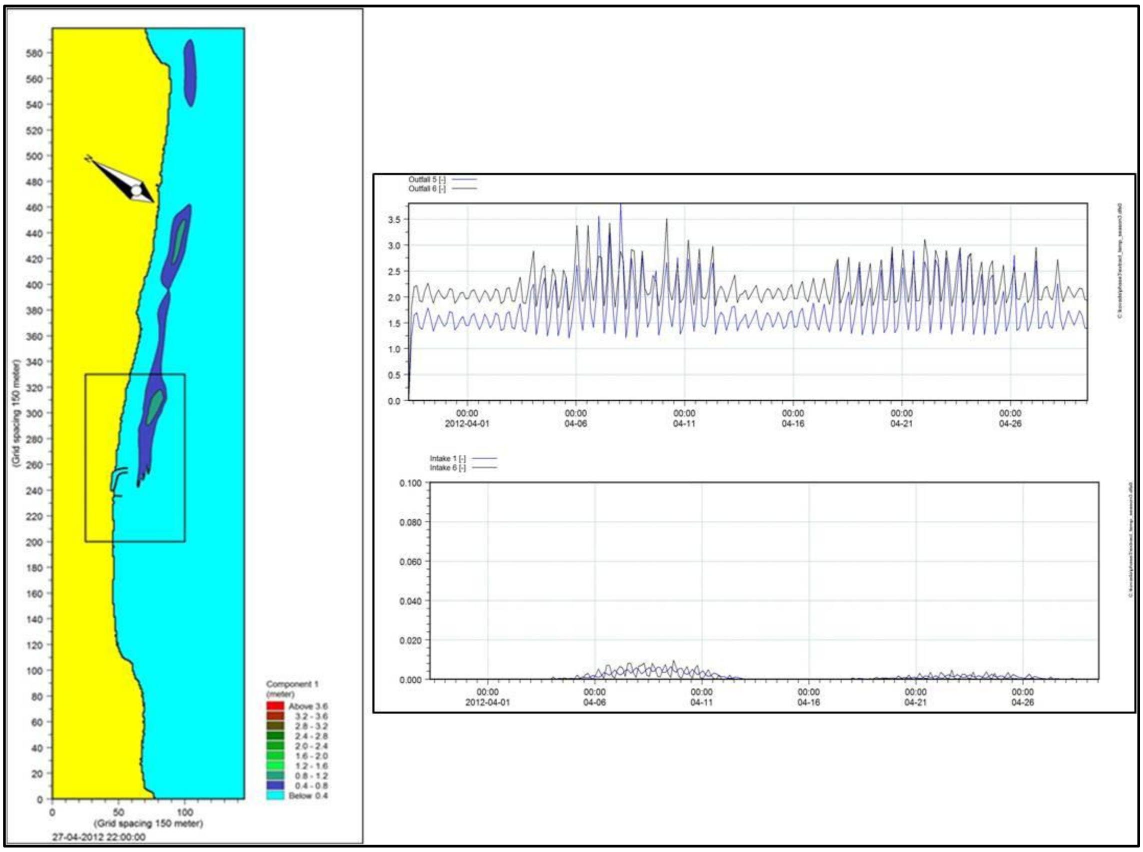


Fig. 19 Model Bathymetry



(a) Thermal plume at surface (b) Time series plot of temperature

Fig. 20 Variation and extent of Thermal plume during critical condition

5.3.2 Three dimensional mathematical model studies

5.3.2.1 *Dispersion of warm water discharge from proposed nuclear power park at Jaitapur, Maharashtra*

Jaitapur is situated in Rajapur Taluka of Ratnagiri District in Maharashtra. It is located on the west coast about 50 km south of Ratnagiri. There is a proposal of Nuclear Power Corporation of India Limited (NPCIL), Mumbai to establish Nuclear Power Plants (NPP) of total installed capacity 1650x6 MWe at Jaitapur. The condenser cooling water required for the power plants is proposed to be drawn from the sea and in turn the warm water will be discharged back to the sea at suitable locations. The cooling water requirement of each unit will be about 110 m^3/s and the rise in water temperature across the condenser will be 7^0 C. The proposed layout of intakes and outfalls consists of six intakes near the shore in the intake channel, which opens towards the north and six outfalls located in the deep sea with outfall O1 and O2 at about 1.5 km from the shore/breakwater, O3 and O4 at about 2 km from the shore/breakwater and O5 and O6 at about 2.5 km from the shore/ breakwater [8]. Layout of intakes and outfalls in model bathymetry is shown in Figure 21. 3D mathematical model studies were carried out to examine the dispersion of warm water discharged from the proposed NPPs and to determine suitability of the proposed layout of intake and outfalls for the condition of minimum recirculation at the intakes considering all the six units are in operation. 3D mathematical modelling software MIKE 3 was used for the simulations. The model was operated for various current conditions prevailing at the site. The studies indicate that the maximum rise in water temperature at the outfall locations would be within 5.0 ^{0}C and at the intakes, it would be within 1.5 ^{0}C. In the creeks the rise in temperature of water would be less than 0.8 ^{0}C (see Figure 22). The rise in water temperature is with respect to the initial water temperature at the intake locations. The maximum rise in water temperature in most regions of the proposed site is observed during transition phase of the currents, which occurs for a short period of time and is not expected to affect the operating efficiency of the power plants significantly. It is further confirmed by the 3D mathematical model studies that the proposed layout is suitable for the condition of minimum recirculation at the intakes [8].

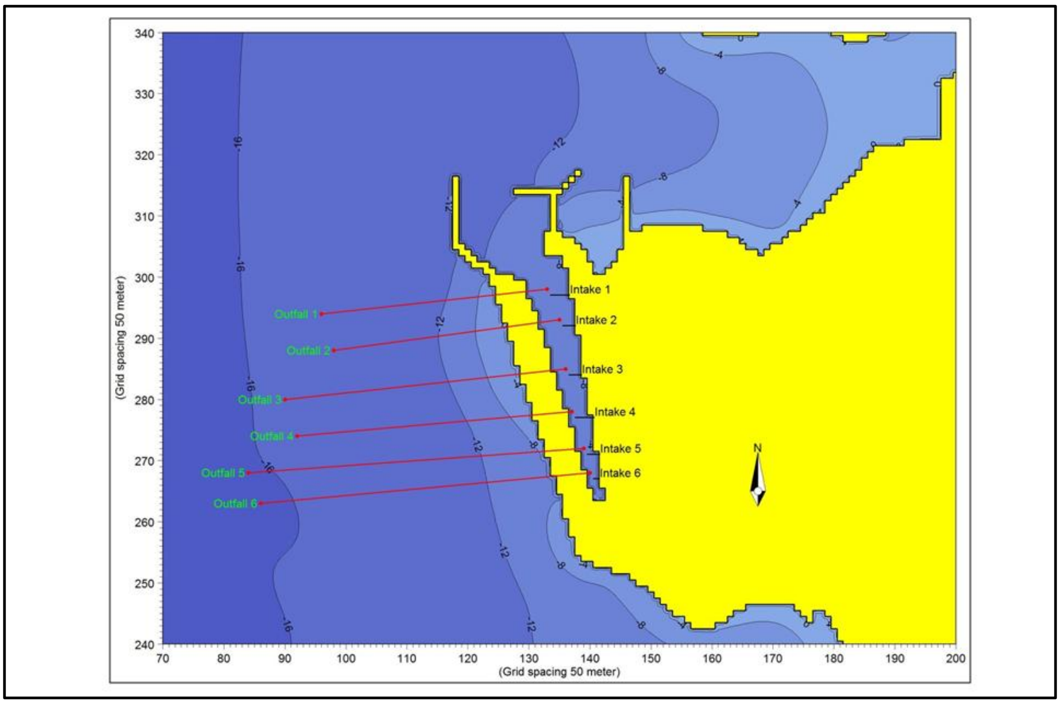
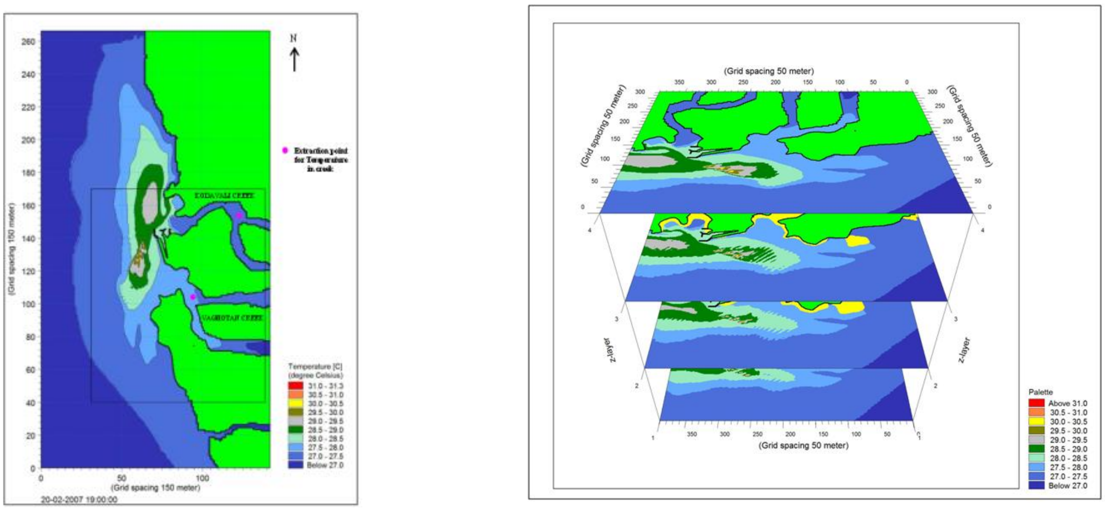


Fig. 21 Part of model bathymetry with six units of intake-outfall



(a) Surface plot (b) Slice plot

Fig. 22 Temperature field for transition phase with all units in operation

5.3.2.2 *Dispersion of warm water discharge from proposed nuclear power park at Kudankulam, Tamil Nadu*

Kudankulam Nuclear Power Park (KKNPP) is located at Kudankulam (Lat 8^0 09' 52" N, Long 77^0 42' 41" E) about 650km south of Chennai in Tirunelveli district of Tamil Nadu, India. Presently, two units of 1000 MWe pressurised water reactor (VVER-1000 PWR) are in commercial operation in phase one of the project. It is proposed to enhance the capability of the project up to 6000 MWe by adding four more units each having 1000 MWe power generation capacity. Model Bathymetry showing intakes and outfalls units is depicted in Figure 23. Similar to the existing power units, it is proposed that cooling water required for condensers of the proposed four units of the power park be drawn from open sea and the warmed seawater would be discharged back to sea at near-shore locations. The cooling water requirement of each unit would be about 85.0 m^3/s and the allowable rise in water temperature across the condensers would be 7 ^0C. The layout of the existing and the proposed power units include intakes structures at open sea, outfalls locations at seashore, and a breakwater structure separating the intakes and outfalls locations for the power park at Kudankulam. 3D mathematical model studies were carried out to examine dispersion of warm water discharge from the NPP and to determine suitability of the existing and the proposed layout of intakes, outfalls and breakwater structure with minimum recirculation in intakes. The 3D mathematical modelling software, MIKE 3 was used for the simulations with the assumption of connected intakes and outfalls. The model was operated for the various current and wind conditions prevailing at site. The studies indicated that the maximum rise in temperature at outfalls location would be within 7.8 ^0C and at the intakes it would be within 0.8 ^0C when the current is eastward. For rest of the period, rise in seawater temperature at intakes would be upto 0.4 ^0C and at the outfalls it would be upto7.5 ^0C above the ambient. Predicted water temperature in excess in 3D view and at surface is shown in Figure 24 and Figure 25, respectively. The rise in water temperature is with respect to initial water temperature at intakes location. The maximum rise in water temperature in most part of the proposed site is observed for a short period of time which will not affect the recirculation at intakes significantly. Hence, it is confirmed by the 3D model studies that the layout along with the proposed expansion of power units would be suitable for the minimum recirculation in the intakes, i.e., efficiency of power generation capacity may be maintained [10].

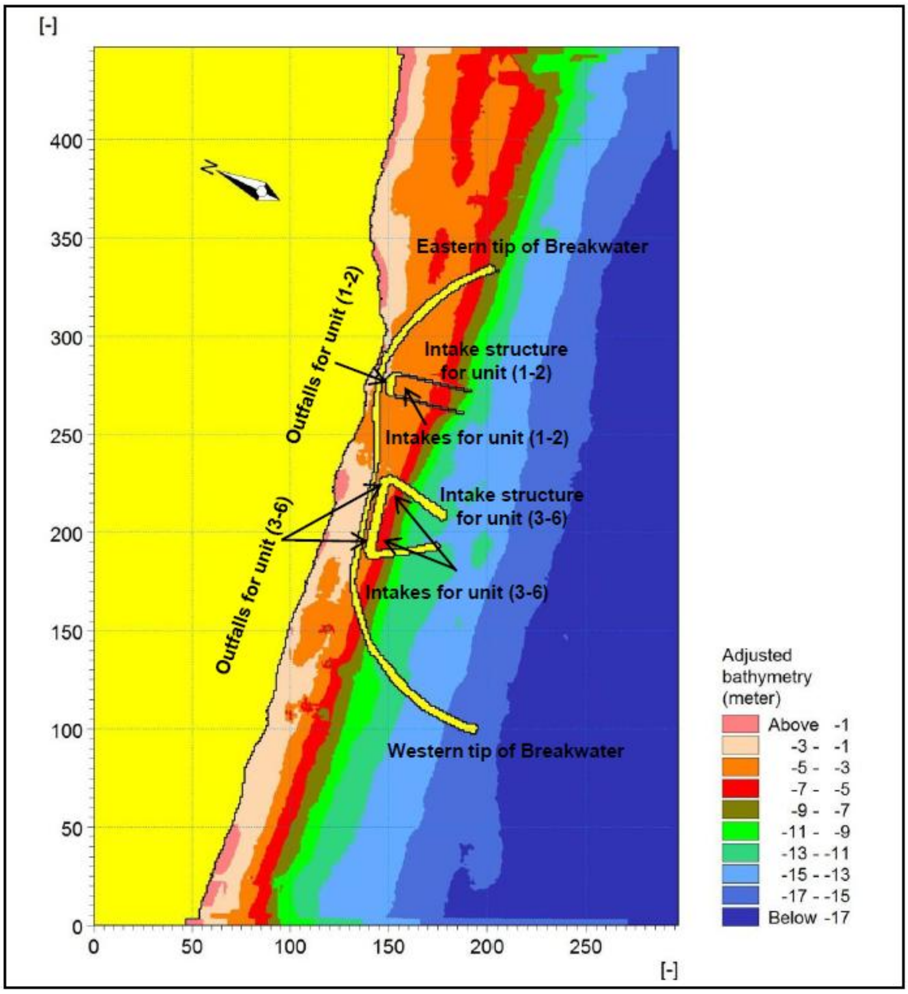


Fig. 23 Part of Model Bathymetry showing intake and outfall

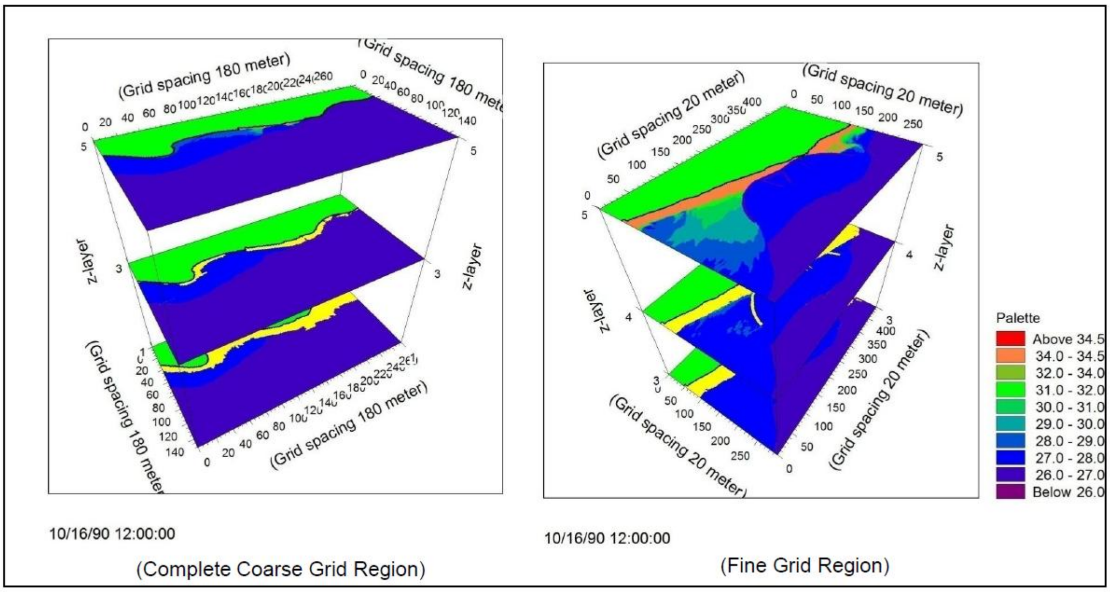


Fig. 24 Predicted Temperature Field in 3-D view

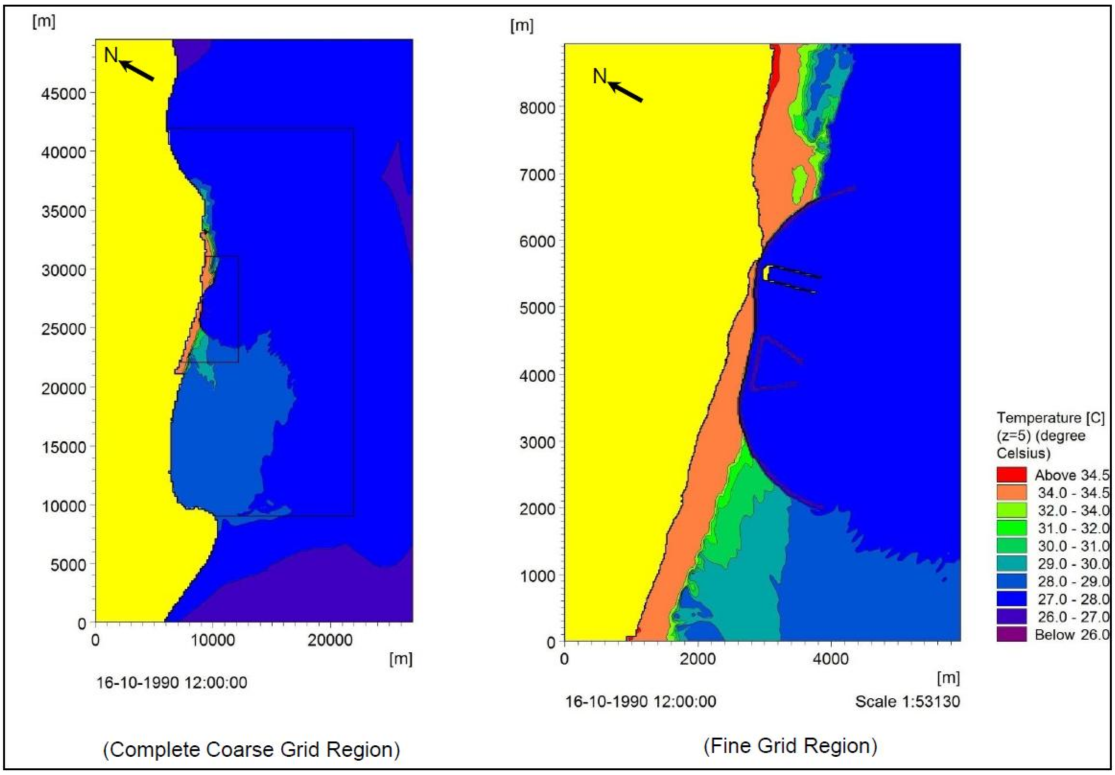


Fig. 25 Predicted Temperature at Surface

Chapter 6

CONCLUSIONS

The growing demand for energy to support and improve the quality of life throughout the world, coupled with the limitation of the world's resources of conventional fuels such as coal and oil, has required the development of alternative sources for electricity supply. The nuclear power plants of today have a considerable thermal efficiency. However, the balance of this energy must be discharged to the environment, and therefore the management of this waste heat to provide the minimum impairment to environmental quality is of great importance, together with the advance of reactor technology to improve the thermal efficiency and so reduce the quantity of heat for disposal. Considerable work has been done in the industrialized countries in regard both to waste heat management and environmental effects of thermal discharges. The discharge of thermal effluent from power plants into water bodies would result in harmful impacts on the ecological life. The higher forms of aquatic life require oxygen for survival and the high temperature decreases its concentration in the water.

The undesirable temperature of effluent water from thermal or nuclear power plant is a concern common to people, industrialists, environmentalists, regulatory agencies etc inhabited in the vicinity of the plant. Secondly, if the warm water discharged through the outfall of power plant enters the intake, it leads to loss of efficiency of the power plant. Coastal processes and the related morphological evolution can be described mathematically by solving partial differential equations, formulated in variables such as velocity, pressure and surface elevation. These differential equations cannot be solved analytically. Numerical models transform these general equation into numerical algorithms which are solved efficiently using computers.

Hydro-thermal modelling of the transport and dispersion of the effluents from the thermal power plants is a powerful tool to predict the trajectory of the thermal plumes and increased water temperature above the ambient. Numerical flow models can be used in the far field modelling in which the inertia force is dominant over the bouncy force. However, numerical models are not capable of predicting the thermal plume in the near field region in which the buoyant force is dominant the over inertia force. This is because physical models are very efficient and reliable tools in studying the multiple complex flow pro- cesses associated with the effluent discharge in the near field region [26]. Thus, physical scale model is a powerful tool to predict the transport and dispersion of the thermal plumes of power plants in the near field region. The results of the physical models enable the prediction of the increased water temperature in the plant vicinity.

The recommended proposal of thermal power plant for NCTPS and ETPS wherein location of intake was placed in Ennore port basin and outfall was located at the mouth of the creek. The outcome of the model runs suggested that the temperature of the hot water at the outlet was 5°C above the ambient. At low water level, the warm water flows towards the ETPS and the drop in temperature was observed to be from 5 °C near the outfall, to 0.5 °C near Railway Bridge over Korttalaiyar backwaters over a distance of about 2.0 km. The temperature drop near the existing intake was observed 1 °C during the low water level, and 0.5 °C during high water level.

Since last three decades mathematical modelling has become a very efficient tool for studying coastal engineering problem. These models are essentially based on conservation of mass and momentum equation. Similar to scale model, there are tidal model as well as wave model. These models can be three dimensional (3D) or two dimensional (2D) depending upon governing equation.

Numerical model software, MIKE 21 AD simulates dispersion of warm water in coastal environment. The modelling software is developed by Danish Hydraulics Institute (DHI), Water and Environment, Denmark and is one of the widely used commercial software It is based on the non-linear vertically integrated 2-D equation of conservation of mass, which takes into account advection and dispersion.

Mathematical modelling software, MIKE 3 is used for three dimensional simulations of coastal hydrodynamics and dispersion of warm water in the study area. This modelling software is also developed by DHI. In order to assess the circulation pattern of warm water, hydrodynamic modelling (MIKE 3 HD) and to determine movement of thermal plume in three dimension , Advection-Dispersion modelling (MIKE 3 AD) is used. Both the modules are included in MIKE 3 model.

There are various types of data such as waves, tides, currents and meteorological data like wind, ambient water temperature, relative humidity, cloud cover, solar insolation etc. which are taken into account for thermal dispersion studies. As the MIKE model relies heavily on the data of climatic factors, it is necessary to take due care in the field collection of data. Further, continuous monitoring of the water body temperature and climatic data is helpful in tuning of the developed model. The model can successfully predict the results only and only if the data which is being used is precise and accurate. Since the data plays an important role in calibration and validation of the model and also impacts various other parameters directly or indirectly, sensitivity analysis if the data is absolutely necessary.

CWPRS has carried out many projects of national importance for thermal dispersion. In mathematical model studies for location of intake / outfall for PFBR at Kalpakkam, thermal recirculation study indicated that combined outflow channel with guided bund of length 680 m from PFBR outfall to MAPS outfall and with minimum length of 500 m beyond MAPS outfall would be required for minimising recirculation of warm water in the intakes. This minimum channel length of 500 m is also required to keep the channel mouth sufficiently away from the MAPS intake and to avoid possible effect of coastline advancement due to the obstruction of the outflow to the littoral drift. In 3D mathematical model studies for dispersion of warm water discharge from proposed nuclear power park at Jaitapur, Maharashtra, the studies indicated that the maximum rise in water temperature at the outfall locations would be within 5.0 ^{0}C and at the intakes, it would be within 1.5 ^{0}C. In the creeks the rise in temperature of water would be less than 0.8 ^{0}C. In another 3D mathematical model studies for dispersion of warm water discharge from proposed nuclear power park at Kudankulam, Tamil Nadu the studies indicated that the maximum rise in temperature at outfalls location would be within 7.8 ^{0}C and at the intakes it would be within 0.8 ^{0}C when the current is eastward. For rest of the period, rise in seawater temperature at intakes would be upto 0.4 ^{0}C and at the outfalls it would be upto 7.5 ^{0}C above the ambient.

In mathematical model studies for dispersion of warm water discharge from proposed nuclear power plant at Kovvada, the main conclusion of the mathematical model studies was that the proposed layout of intakes, outfalls and inclined intake channel with locations of outfalls in staggered manner was suitable as it did satisfy the criteria of permissible limit of temperature circulation at intakes. However, during the transition phase of currents, which represents transitional phase of tidal currents at the site, the rise in temperature at intakes was about 2.5 ^{0}C above the ambient temperature. However, as the period of occurrence of this phase was relatively less, such a small rise in intake temperature occurring for a short duration might be permissible with the existing location of the power plant.

It is important to note that nuclear power projects are quite sensitive projects as they deal with the radioactive materials. Also, there are many costs involved at each and every stage. Hence, even the smallest of a mistake in the execution of the design at site may lead to a heavy financial and maybe even loss of life. Hence, utmost care needs to be taken while executing the nuclear power plant establishment.

ACKNOWLEDGEMENT

Authors would like to express their sincere thanks to Dr. (Mrs.) V.V. Bhosekar, Director, CWPRS for her constant support and guidance. Her continuous encouragement helped the authors to complete the Technical Memorandum in time.

Authors special thanks are due to Mrs. Vaibhavi Roy, Research Assistant for her consistence support to complete this memorandum. She has taken immense efforts in collecting and compiling the relevant material.

Authors are thankful to Project Authorities viz., Nuclear Power Corporation of India Limited (NPCIL), Indira Gandhi Centre for Atomic Research (IGCAR), Tamil Nadu Electricity Board (TNEB), for providing support and relevant data for the various project works. Authors are also thankful to Shri V.V. Vaze, Chief Research Officer (Retired), Mrs. S.S. Kunte, Chief Research Officer (Retired), Shri S.G. Manjunatha, Scientist D and the team for providing inputs for thermal model studies. Authors are also thankful to Mrs. Shivani Sahu, Scientist B for her help in compilation of case studies of thermal power project for this memorandum.

Authors would also like to thank Officers and staff of WAPIS, CWPRS for extending their help for arranging necessary technical material relevant to this memorandum and for printing.

Last but not least, authors would also like to thank all officers and staff of MMCE division for their kind co-operation, help and motivation during the preparation of this memorandum.

REFERENCES

1. Baek, K.O., Seo, I.W. (2010). "Routing procedures for observed dispersion coefficients in two-dimensional river mixing." Adv. in Wat. Res., v.33(12), pp.1551–1559.

2. Bleninger, T. (2006). "Coupled 3D hydrodynamic models for submarine outfalls: Environmental hydraulic design and control of multiport diffusers". PhD Thesis, Institute for Hydromechanics, University of Karlsruhe, Karlsruhe.

3. CWPRS Technical Report No. 3857 (2002). "Physical model studies for locating intake and outfall structures of cooling water system for North Chennai Thermal Power Station", Mar. 2002, CWPRS, Pune.

4. CWPRS Technical Report No. 3903 (2002). "Field data collection and analysis for condenser cooling water system (CCWS) of PFBR", July 2002, CWPRS, Pune.

5. CWPRS Technical Report No. 4042 (2003). "Mathematical model studies for location of intake /outfall for 500 MWe Prototype Fast Breeder Reactor at Kalpakkam, Tamil Nadu (Revised)", Oct. 2003, CWPRS, Pune.

6. CWPRS Technical Report No. 4329 (2006). "Supplementary Mathematical model studies for littoral drift and thermal recirculation for sea water intake /outfall of Fast Breeder Reactor Project (FBR)", April 2006, CWPRS, Pune.

7. CWPRS Technical Report No. 4918 (2012). "Mathematical model studies for confirming locations of intake and outfall of proposed power plant at Salav, Maharashtra", Jan. 2012, CWPRS, Pune.

8. CWPRS Technical Report No. 5140 (2014). "3D Mathematical model studies for dispersion of warm water discharge from proposed Nuclear Power Park at Jaitapur, Maharashtra", Jan. 2014, CWPRS, Pune.

9. CWPRS Technical Report No. 5318 (2015). "Mathematical model studies to examine thermal dispersion of warm water discharge from Nuclear Power Project at Kovvada, Andhra Pradesh", Sept. 2015, CWPRS, Pune.

10. CWPRS Technical Report No. 5725 (2019). "3D Mathematical model studies for dispersion of warm water discharge from proposed Nuclear Power Park at Kudankulam, Tamil Nadu", July 2019, CWPRS, Pune.

11. Central Electricity Authority 2018. "Growth of electricity sector in India from 1947-2019", Annual Report 2017-18, May 2018. Retrieved 28 Aug. 2019.

12. Cheung, S.K.B., Leung, D., Wang, W., Lee, J.H.W., Cheung, V. (2000). "VISJET-a computer ocean outfall modelling system", Conf. Paper, February 2000, DOI: 10.1109/CGI.2000.852322

13. Daware, K. (2015). Basic layout and working of a thermal power plant, https://www.electricaleasy.com/2015/08/thermal-power-plant.html

14. Daware, K. (2015). Basic layout and working of a nuclear power plant, https://www.electricaleasy.com/2015/09/nuclear-power-plant.html

15. Deas, M. L., Lowney, C. L. (2000), "Water temperature modelling review in central Valley", California Water Modelling Forum, September 2000.

16. Fischer, H.B., List, E.J., Koh, R.C.Y., Imberger, J., Brooks, N.H. (1979). Mixing in inland and coastal waters, Academic Press, San Diego, pp. 104-147.

17. Frick, W.E. (2004). "Visual Plumes mixing zone modeling software", Env. Modeling and Software, v.19 (7-8), pp.645-654.

18. Gualtieri, C.,(2009). "RANS-based simulation of transverse turbulent mixing in a 2D geometry", Env. Fluid Mech., v.10, pp.137–156.

19. International Atomic Energy Agency, 1974. Thermal discharges at nuclear power stations: Their management and environmental impacts. Vienna: Tech. Rept. Ser. No. 155.

20. International Atomic Energy Agency. 2018. Energy, Electricity and Nuclear Power Estimates for the period upto 2050, IAEA, Vienna, Aug. 2018.

21. Jain, S.K. (2010). Nuclear Power in India – Past, Present and Future, https://www.npcil.nic.in/

22. Jha, S.N., Manivanan, R., Saptarishi, P.G. (2010), "Relationship and impact of climatic factors with coastal environmental parameters due to warm water discharge from a proposed power plant", Proc. of the 9th Int. Conf. on Hydro-Science and Engineering, ICHE-2010.

23. Jha, S.N., Manivanan, R., Saptarishi, P.G. (2010), "Development of a mathematical model for determining temperature of a lake based on climatic variations and its validation using measured data", Proc. of 9th Int. Conf. on Hydro-Science and Engineering, ICHE-2010.

24. Jirka, G.H. , Akar, P.J., (1991). "Hydrodynamic classification of submerged multiport diffuser discharges", Jl. of Hyd. Eng., ASCE, v.117, pp.1113-1128.

25. Jirka, G.H., Doneker, R.L., Hinton, S.W., (1996). "User's Manual for CORMIX: A hydrodynamic mixing zone model and decision support system for pollutant discharges into surface waters", U.S. Env. Prot. Agency, Tech. Rep., Environmental Research Lab, Athens, Georgia, USA.

26. Jirka, G. H. (2004). “Integral model for turbulent buoyant jets in unbounded stratified flows. 1: The single round jet.” Env. Fluid Mech., v.4, pp.1–56.

27. Jones, G. R., Nash, J. D., Jirka, G. H. (1996). “Cormix3: An expert system for mixing zone analysis and prediction of buoyant surface discharges.”Technical Rep., De Frees Hydraulics Laboratory, Cornell Univ., Cornell, NY.

28. Jones, G., Nash, J., Doneker, R., Jirka, G. (2007). “Buoyant surface discharges into water bodies. I: Flow classification and prediction methodology.” Jl. Hyd. Eng., v.133(9), pp.1010–1020.

29. Kolfat, T. D. (1971). "Thermal Discharges - An Overview." Proc. of the American Power Conf., 33, pp.412-426.

30. Kudale, M.D. (2012), "Overview of Coastal Engineering", Short Course: Modeling and data needs for coastal engineering projects, CWPRS, Oct. 2012.

31. Lamarsh, J. R. (1983). Introduction to Nuclear Reactor Theory, 2nd ed., Addison-Wesley, Reading, MA (1983).

32. Lamarsh, J. R., Baratta, A. J. (2001), Introduction to nuclear engineering, 3d ed., Prentice-Hall, 2001, ISBN: 0-201-82498-1.

33. Lin, C.Y., Holley, E.R., Maxwell, W.H.C. (1977). “Buoyant surface jets discharged into a strong cross flow”. University of Illinois at Urbana-Champaign. Final report, Project No. B-088-ILL.

34. Manjunatha, S.G., Bobade, K.B., Kudale, M.D. (2015), "Pre-cooling technique for a thermal discharge from coastal thermal power plant", Procedia Engineering, V. 116, pp. 358-365.

35. MIKE 21 manual (2014), DHI, Denmark.

36. MIKE 3 manual (2007), DHI, Denmark.

37. MoEF, EP Rule 1986.

38. Mumbai Mirror (2013). Kudankulam nuclear plant begins power generation, 22 October 2013. Retrieved 29 January 2014.

39. Muralikrishna, I.V., Manickam, V. (2017). Environmental management: Science and engineering for industry, B.S. Publication, India. Elsevier.

40. Nekouee, N., Roberts, P. J. W., Schwab, D. J., McCormick, M. J. (2013). “Classification of buoyant river plumes from large aspect ratio channels. ” Jl. Hyd. Eng., v. 139(3), pp.296–309.

41. NRDC Symposium. (1974). "Report on symposium: Environmental effects of cooling systems at Nuclear power plants, Proceedings of a symposium on the physical and biological effects on the environment of cooling systems and thermal discharges at nuclear power stations", Norway, Aug. 1974.

42. NRDC issue brief. (2014). "Power plant cooling and associated impacts: The need to modernize U.S. power plants and protect our water resources and aquatic ecosystems", IB:14-04-C. April 2014.

43. Palomar, P., Lara, J., Losada, I., Rodrigo, M., and Alvrez, A. (2012). "Near field brine discharge modelling part 1: Analysis of commercial tools. Desalination", v.290, pp.14–27.

44. Pilechi, A. (2016). "Numerical modelling and field study of thermal plume dispersion in rivers and coastal waters", PhD Thesis, Dept. Civil Eng., Univ. of Ottawa, Canada.

45. Policastro, A. J. (1972). "State-of-the-art of surface thermal plume modelling for large lakes", Argonne National Laboratory Annual Meeting of the American Institute of Chemical Engineers, Proceedings. New York.

46. Purohit, A. A. (2012), "Physical Tidal Modeling – Case Studies", Short Course: Modeling and Data needs for Coastal Engineering Projects, CWPRS, Pune, Oct. 2012.

47. Roberts, P.J.W. (2013). “Ocean Outfalls.” In handbook of Environmental Fluid Dynamics, Vol. 2, H. J. Fernando, Ed., CRC Press: 229-242.

48. Rutherford, J.C. (1994). River mixing. Wiley, Chichester (England).

49. Shah, V., Dekhatwala, A., Banerjee, J. and Patra, A. K. (2017), "Analysis of dispersion of heated effluent from power plant: a case study", Sadhana, V. 42, No. 4, pp. 557–574.

50. Sinha, J. (2012), "Mathematical modelling techniques for thermal circulation", Short Course: Modelling and Data needs for Coastal Engineering Projects, CWPRS, Oct. 2012.

51. Thomann, R. V., Mueller, J. A. (1987), "Principles of surface water quality modelling and control", Harper & Row, 1987.

52. www.world-nuclear.org/information-library/country-profiles/countries-g-n/india.aspx (2019). Nuclear Power in India.

VISION

To be a world-class centre of excellence in hydraulic engineering research and allied areas; which is responsive to changing global scenario, and need for sustaining and enhancing excellence in providing technological solutions for optimal and safe design of water resources structures.

MISSION

- To meet the country's need for basic & applied research in water resources, power sector and coastal engineering with world-class standards
- To develop competence in deployment of latest technologies by networking with the top institutions globally, to meet the future needs for development of water resources projects in the country effectively
- To disseminate information, build skills and knowledge for capacity-building and mass awareness for optimization of available water resources

MAJOR FUNCTIONS

- Undertaking specific research studies relating to development of water resources, power and coastal projects
- Consultancy and advisory services to Central and State Governments, private sector and other countries
- Disseminating research findings and promoting/assisting research activities in other organizations concerned with water resources projects
- Contributions to Bureau of Indian Standards and International Standards Organization
- Carrying out basic and applied research to support the specific studies
- Contribution towards advancements in technology through participation in various committees at National and State Levels

Central Water and Power Research Station
Khadakwasala, Sinhagad Road, Pune 411 024

Telephone : +91-20-24103200/ 24381801
Fax : +91-20-24381004
Web : www.cwprs.gov.in