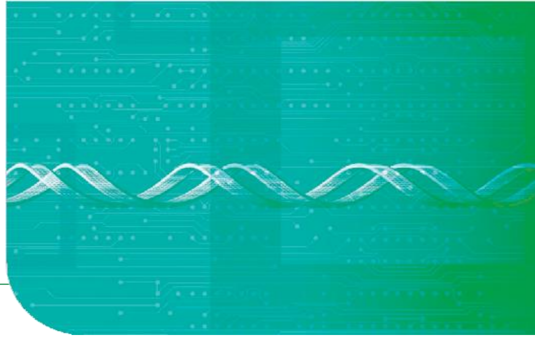
Analysis Example

Water Treatment Solutions Using Ansys Software and Flownex



1 Introduction

Simulation and CAE will play an increasingly important role in product development as the digitalization of product development becomes more prevalent. In particular, it is noteworthy that the use of more in-depth simulation and CAE is increasing, as products are required to have higher performance in order to achieve the SDGs and energy conservation that are being actively advocated these days. In the field of water treatment, CAE technology is also used in a variety of processes, from initial facility design and verification of facility operation to risk management. This paper focuses on fluid analysis, a field of CAE, and introduces various examples using Ansys® CFD™, a fluid simulation tool from Ansys, and Flownex®, a 1D simulation tool for analyzing entire systems from M-tech.

2 Introduction of a case study of 3D simulation related to water treatment using Ansys CFD

Example of simulation application in activated sludge water treatment

In recent years, simulation has been increasingly used to improve the efficiency of water treatment plants. The following is a list of the sewage inflow, sedimentation tank, and aeration tank (aeration).

Figure 1 shows the simulation of the flow in the inflow section of sewage and the retention time of solids in the sewage. Simulation can be used to analyze pipe layout for the most efficient flow possible, as well as the retention time of solids mixed in sewage, to ensure that solids do not remain where they are.

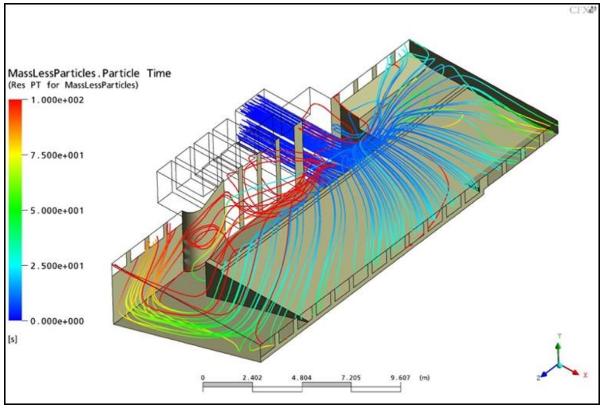


Figure 1. Solid retention time in sewage inflow

Figure 2 shows an example of analysis of a sedimentation tank. The red area indicates the distribution of sludge. The optimal depth and number of sedimentation tanks can be considered for efficient water separation.

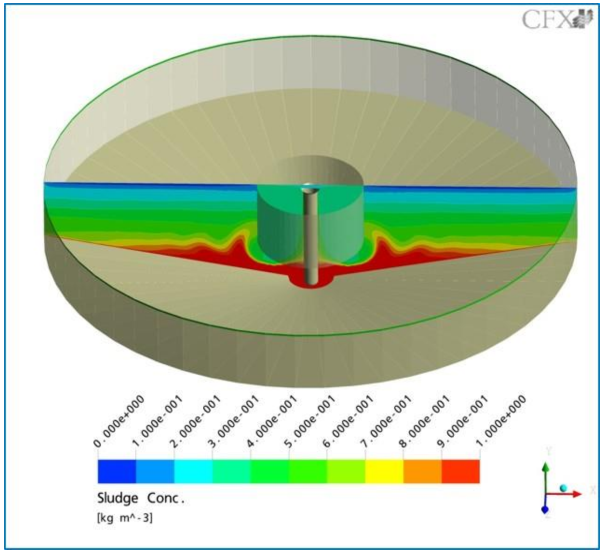


Figure 2. Sludge distribution in sedimentation tank

Figure 3 shows an example analysis of an aeration tank. Oxygen can be fed into the liquid to determine whether oxygen is distributed throughout the entire liquid. The analysis can include not only the liquid phase but also the gas phase, and it is possible to model the dissolution of oxygen into the water.

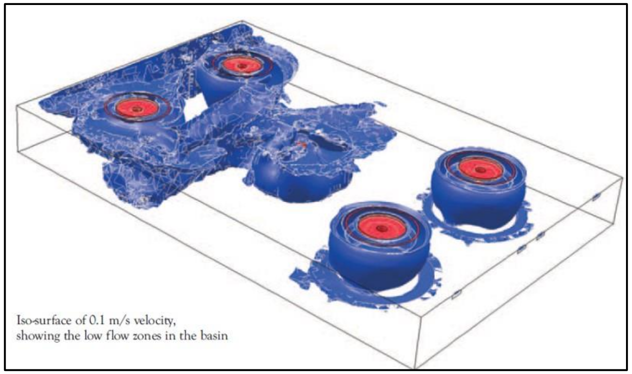


Figure 3. Oxygen distribution in the aeration tank

Thus, it is possible to analyze each process of a sewage treatment facility, and these simulations prove to be useful in aiming for higher efficiency and energy conservation.

3 Introduction of 1D system simulation case study using Flownex

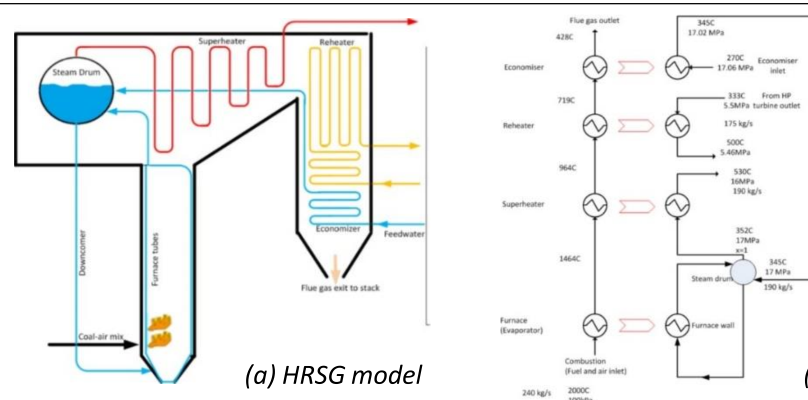
3.1 Introduction of thermal fluid system simulation software, Flownex SE

In recent years, demand for CO^2 reduction and energy conservation have led to calls for higher efficiency throughout the equipment. To achieve these goals, 3D CFD and CAE, as shown in the above examples, are being used, but they are often limited to partial analysis of a single piece of equipment. On the other hand, understanding and optimizing the performance of the entire equipment is also essential to achieving overall high efficiency, and there are useful system analysis tools available to perform these tasks. This chapter will introduce Flownex, a simulation software (1D simulation), and examples of its simulations. Flownex is a thermal fluid system simulation software that can analyze fluid machinery (pumps, turbines, etc.), boilers, and other equipment, as well as large-scale systems that combine them, such as power plants, pipe systems, and air conditioning systems.

3.2 1D System Simulation Case Study

3.2.1 Example of exhaust heat recovery boiler system analysis

This section presents an example of a system simulation of a heat recovery steam generator (HRGS). Boilers are widely used as heat sources not only in power generation plants but also in buildings, apartments, and hotels. A boiler consists of a combustion furnace and a heat exchanger. The physical phenomena include complex physical phenomena such as combustion (chemical reactions), high temperature and pressure flow, phase changes of water and steam, and heat transfer (heat conduction, heat transfer, and radiation). Since it is impractical to perform a 3D analysis of the entire boiler taking these phenomena into account, it is effective to use system simulation software such as Flownex to analyze the boiler. Figures 4(a) and 4(b) provide a schematic and diagram of the boiler. It consists of four sections: Furnace/Evaporator to heat and evaporate the steam, Superheater to heat the steam to the high pressure turbine, and Re-heater to reheat the steam discharged from the high pressure turbine.



(a) HRSG model

(b) HRSG diagram

Figure 4. HRSG system simulation [(a) and (b)].

Figure 4(c) displays the temperature distribution in the boiler network system. It can be seen that the temperature of the combustion exhaust gas decreases as it moves downstream and the temperature of the water/steam increases as it moves downstream. Figures 4(d) and 4(e)show the water level of the steam drum and the temperature of each section over time, indicating that the temperature decreases with time and the amount of condensation increases accordingly, causing the water level of the steam drum to rise. Thus, thermo-fluid system simulation software makes it possible to check the behavior of the entire system.

3.2.2 Example of analysis of irrigation canal network

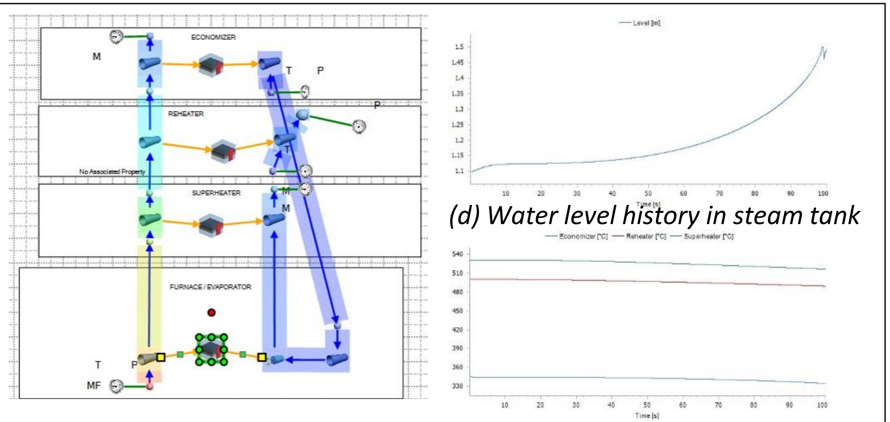
Figure 5 shows an example of a simulation of an irrigation canal network. The purpose of the simulation is to design the irrigation canal network so that water can be pumped evenly to the supply sites. The contour plot shows the flow rate in each channel, and it can be seen that the initial design shown in Figure 5(a) does not deliver water evenly, and the flow rate is smaller downstream. Based on this, optimization was performed by adding a valve to the flow contour after optimization as shown in Figure 5(b).

By adding valves and optimizing their openings, we are able to equalize the flow rate at all supply points. Thus, thermal fluid system simulation software allows optimization of the entire system to be studied in the initial design stage.

3.2.3 Example of analysis of a pump station for fire protection equipment

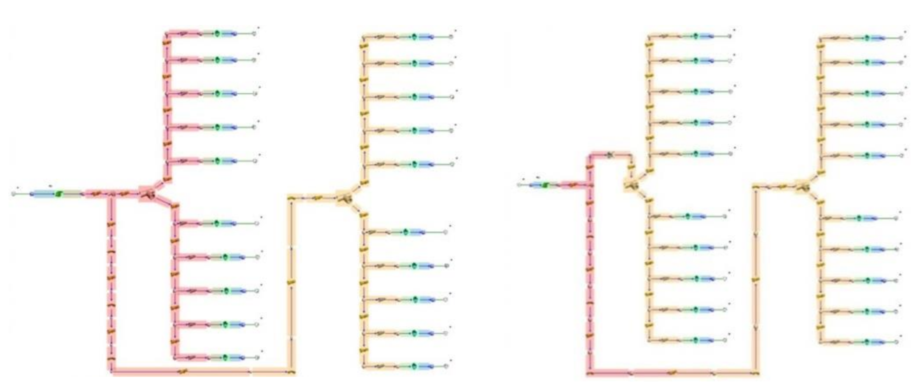
Fire protection systems are installed in power plants, petrochemical plants, warehouses, etc. for the purpose of fire suppression or fire prevention. Although the frequency of use of these fire protection systems is limited, they are an integral part of the safe operation of many power plants. In this example analysis, the fire safety of a boiler head in a coal-fired power plant.

This section describes simulations assuming startup conditions for the entire system. Excessive pressure fluctuations can be a problem in fire suppression



(c) HRSG network model *(d) Water level history in steam tank* *(e) Temperature history*

Figure 4. HRSG system simulation [(c)-(e)]



(a) Before optimization *(b) After optimization*

Figure 5. Example of irrigation canal network

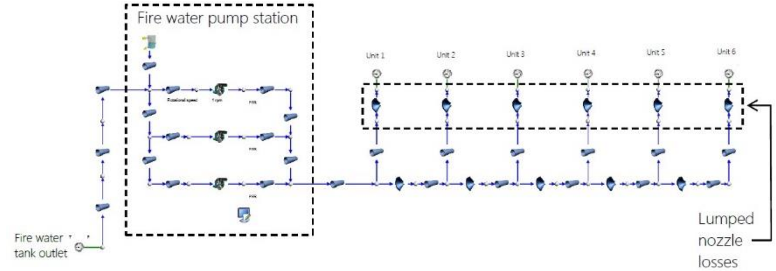
feed water systems at coal-fired power plants during pump startup, and the purpose of this simulation is to study the avoidance of pressure fluctuations. In this system simulation, pump performance was specified as a graph of pressure rise versus volumetric flow rate, and electric motor behavior was modeled in terms of torque curve and moment of inertia values. Assuming moment of inertia values of 1.31 kg.m^2 and 11.78 kg.m^2, these two moment of inertia values are used in the simulation. The water level of the main fire extinguishing tank should be 3.6 m.

The following cases were considered

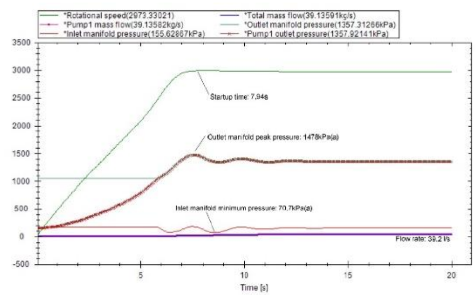
1. Starting with only one pump (moment of inertia 11.78 kg.m^2)
2. Starting with only one pump (moment of inertia 1.31 kg.m^2)
3. Start a second pump while one pump is already running

Figure 6 shows the present analytical network model and the simulation results obtained in the aforementioned cases.

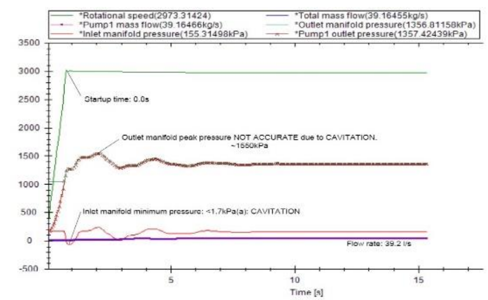
Figures 6(b) and 6(c) show that the time to reach the specified flow rate is longer for a single pump with a full-voltage starting moment of inertia of 11.78 kg.m^2 and shorter for a single pump with a moment of inertia of 1.31 kg.m^2, but that large pressure fluctuations occur in the early startup phase. This is caused by rapid acceleration of the pump, leading to pressure fluctuations in the piping system. In the case of the pump, the



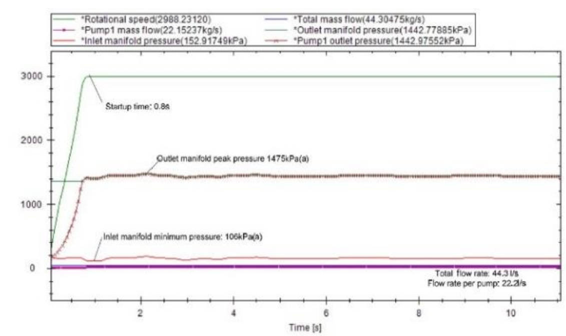
(a) Fire protection equipment pump station network model



(b) Starting with only one pump (moment of inertia = 11.78 kg.m^2)



(c) Starting with only one pump (moment of inertia = 1.31 kg.m^2)



(d) Starting a second pump when one pump is already running. (moment of inertia = 1.31 kg.m^2)

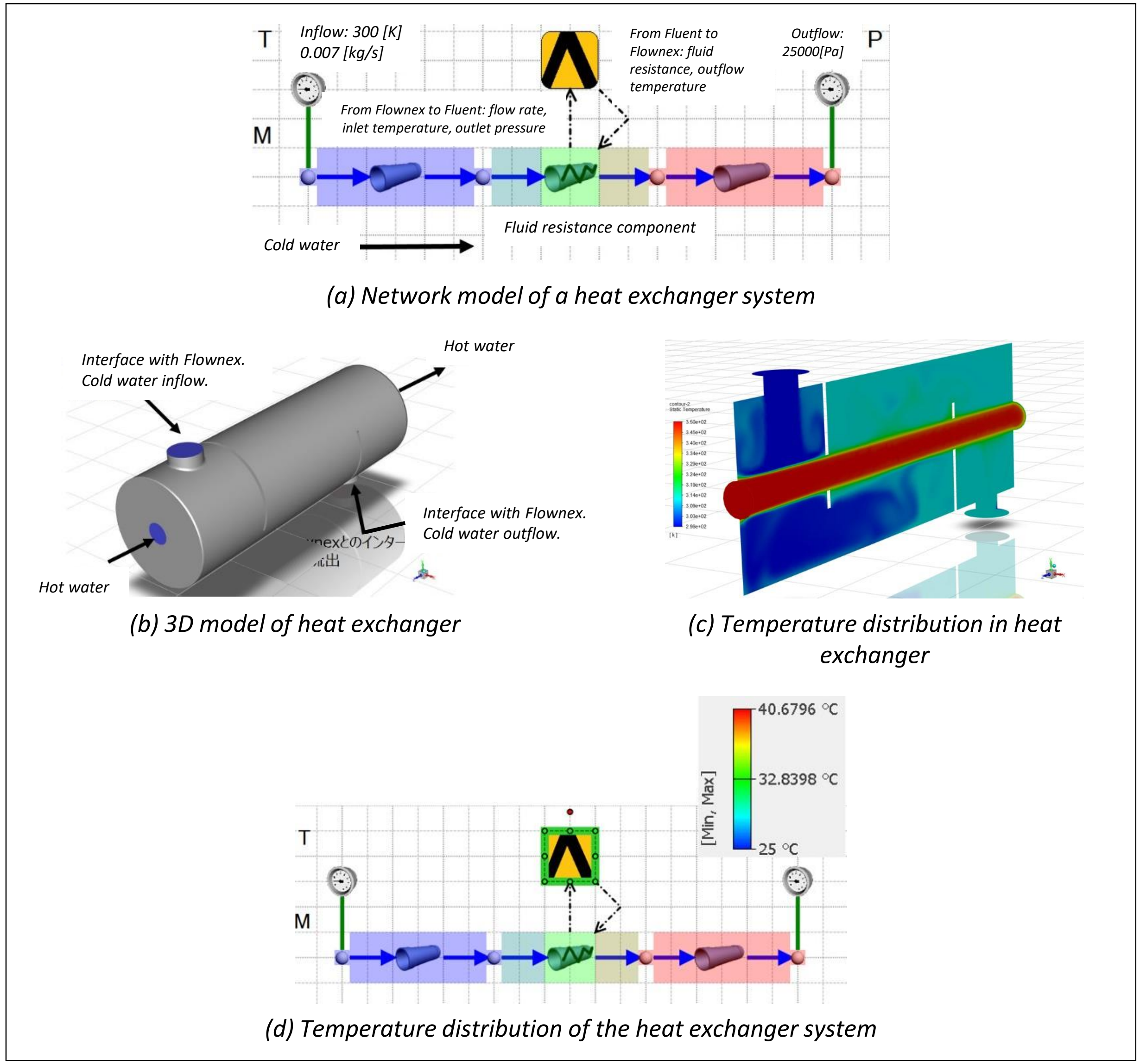
Figure 6. Fire protection equipment pump station

pressure drop at the inlet is below the saturation pressure, causing cavitation to occur in the pump. Figure 6(d) shows the results when "starting the second pump while one pump is already running," indicating that the time to reach the specified flow rate is short and pressure fluctuations are suppressed. By simulating with Flownex in this way, we were able to confirm in advance the behavior of the pump during startup in each case and found the most effective starting conditions. While these studies are difficult to perform in a 3D simulation at a realistic computational cost, they can be performed in a relatively short time by conducting a system analysis using a 1D system simulation software such as Flownex.

3.2.4 Coupled analysis of 3D simulation and 1D system simulation (heat exchanger)

The following case study is a combination of 3D simulation and system simulation. In this case study, the heat exchanger section is simulated in 3D using Ansys Fluent®, and the other piping is simulated in 1D system simulation using Flownex. Since using 3D simulation for everything increases the computational cost, it is possible to replace some of the simulation with 1D system simulation, which is more efficient and accurate.

Figure 7 shows the network model and the 3D model used in this analysis, and simulation results. The flow, temperature, and pressure on the upstream of the piping are transferred from Flownex (system simulation side) to Ansys Fluent (3D simulation side), and the temperature



(a) Network model of a heat exchanger system

(b) 3D model of heat exchanger

(c) Temperature distribution in heat exchanger

(d) Temperature distribution of the heat exchanger system

Figure 7. Co-simulation of 3D and 1D systems

and pressure drop after flow through the heat exchanger are transferred from Ansys Fluent (on the 3D simulation side), thereby achieving a coupled analysis of 3D simulation and 1D system This enables coupled analysis of 3D simulation and 1D system simulation. This coupled analysis can be used to study the detailed design of the heat exchanger, taking into account the behavior of the entire system. 1D system simulation. This coupled analysis can be used to study the detailed design of the heat exchanger, taking into account the behavior of the entire system.

4 Conclusion

This paper focuses on simulation cases using Ansys CFD, a 3D fluid analysis tool from Ansys, and Flownex, a 1D simulation tool from M-tech, and presents examples of 3D simulation applications for activated sludge process water treatment and 1D system simulations for boilers, irrigation channels, pump stations and 1D system simulations for boilers, irrigation channels, pump stations, and heat exchangers.

3D simulation and 1D system simulation technologies are expected to continue to develop, and technological development combined with AI (machine learning and deep running) and IoT is gaining momentum. In addition to front-row fielding in facility design, simulation is also being used to monitor operating conditions and predict changes in conditions during facility operation, a technology known as "digital twin. We hope that the case studies introduced in this issue will help readers of this magazine to solve their business issues.

This document was first published by CYBERNET SYSTEMS CO.,LTD., Japan (Manufacturing with CAE Vol.38)

Inquiries

CYBERNET SYSTEMS MALAYSIA SDN.BHD.

Phone number: +60(3) - 2201-1221
(日本語で対応可能です)
Email: information@cybernet.asia
Website: www.cybernet.asia/ansyscfd