9-28-2015 12:00 AM

Modeling, Construction, and Validation of a Simulator for a Nuclear Process Control Test Facility

Binggang Cui, *The University of Western Ontario*

Supervisor: Jin Jiang, *The University of Western Ontario*

A thesis submitted in partial fulfillment of the requirements for the Master of Engineering Science degree in Electrical and Computer Engineering

MODELING, CONSTRUCTION, AND VALIDATION OF A SIMULATOR FOR A NUCLEAR PROCESS CONTROL TEST FACILITY

(Thesis format: Monograph)

by

Binggang Cui

Graduate Program in Electrical and Computer Engineering

A thesis submitted in partial fulfillment
of the requirements for the degree of
Master of Engineering Science

The School of Graduate and Postdoctoral Studies
The University of Western Ontario
London, Ontario, Canada

Abstract

This thesis presents the modeling, construction, and validation of a software simulator for a facility with physical components, referred to as NPCTF (Nuclear Process Control Test Facility). The simulator focuses simulation functionalities on dynamic behaviours of thermal-hydraulic systems related to nuclear power plants.

The modeling techniques adopted to construct the simulator are based on physical principles. Dynamic models are developed to reproduce the dynamic characteristics of NPCTF. Based on these mathematical relationships, computer codes are developed with the toolbox Simscape. The simulator can be used to investigate specific scenarios of NPCTF under similar operating conditions.

A simulation environment, the toolbox Simscape in Simulink of Matlab, herein, is selected as the platform to set up the simulator. With the features of a physical network approach and built-in component library in Simscape as well as custom-defined blocks created upon the formed dynamic models, the simulator is constructed by integrating component blocks into (i) primary coolant loop, (ii) secondary water loop, (iii) pressurized air loop, and (iv) some auxiliary loops within physical domains. The integration of components forms a mathematical matrix to represent underlying principles of the NPCTF. By employing an implicit fixed-step solver and identifying appropriate parameters of the global configuration and component blocks, the simulator can mimic the responses of NPCTF, under similar conditions.

Specifications of the simulator are defined. And the configuration and implementation scheme are developed for the simulator. Through verification and validation, it can be concluded that the developed simulator can be used in conjunction with NPCTF to support instrumentation and control system research for nuclear power plants.

Keywords: Software simulator, Nuclear power plant process, Modeling technique, Physical principles, Dynamic model, Simscape

Acknowledgments

I would like to express my sincerest gratitude to my supervisor Dr. Jin Jiang firstly, not only for his offering the opportunity for me to further my study in the amazing university, but also for his guidance, patience, and continuous help throughout the entire duration of the research work. Without his appreciation, I could not come back to the campus to further my academic work.

I must thank Dr. Xinhong Huang for her management of research and thesis processes as well as her suggestions in improving the thesis quality and my writing skill. I also wish to thank all the members of the Control, Instrumentation and Electrical Systems (CIES) research group for their supports in many ways.

Thanks also go to the Natural Science and Engineering Research Council of Canada (NSERC) and the University Network of Excellence in Nuclear Engineering (UNENE) for their financial supports of this work.

The greatest gratitude I wish to express to my wife Lu Zheng, for her encouragement, understanding, and support. I also thank my daughter and son, Aiying Cui and Evan Cui, for their happiness and loves filling my life.

Table of Contents

List of Tables

List of Figures

Abbreviation and Nomenclature

t	calculation step, s
u	velocity, m/s
h	convective heater transfer coefficient, $W/(m^2\ K)$
k	thermal conductivity, $W/(m\ K)$
g	gravity acceleration, N/kg *or* m/s^2, g=9.80665 N/kg
p	pressure, *Pa(Pascal)*, $1Pa=1.450377\times10^{-4}$ *PSI(Pounds per Square Inch)*
Cp	specific heat at constant pressure, $J/(kg\ K)$
T	thermodynamic temperature, K
F	mass flow rate of water or air, kg/s
G	volumetric flow rate of water or air, m^3/s
Q	heat transfer rate, J/s
q	heat flux, W/m^2
L	water level, m
P	power, W
U	voltage, *V (Volt)*
I	current, *A (Ampere)*
V	volume, m^3, 1 m^3=1000*l(liter)*, *1 US gallon=3.78541 liter*
m	mass, kg

H	height, *m, 1m=39.370 in(inch)*
l	length, *m*
D	diameter, *m*
A	area, m^2
P_{per}	perimeter, *m*
J_ω	moment of inertia, $kg\ m^2$
R	hydraulic resistance or pneumatic resistance
x, y, z	space coordinates in Cartesian system
Nu	Nusselt number
Re	Reynolds number
Pr	Prandtl number

Greek symbols

ρ	density, kg/m^3
α	thermal diffusivity, m^2/s
μ	dynamic viscosity, *kg/(m s),or Pa s, or N s/* m^2
ν	kinematic viscosity, m^2/s
ω	angular velocity, *rad/s*

Subscripts

w	water
m	metal
a	air
e	electricity
i	inlet or inner
o	outlet or outer
l	lost heat, or laminar flow
t	turbulent flow
0	initial condition
1	specially denotes primary side
2	specially denotes secondary side
amb	ambience
htr	heater
clr	chiller
shh	sheath of electrical heating rod

Abbreviations

NPCTF: Nuclear Process Control Test Facility

NPP: Nuclear Power Plant

I&C: Instrumentation and Control

V&V: Verification and Validation

P&ID: Piping and Instrumentation Diagram

DAE: Differential-Algebraic Equation

ODE: Ordinary Differential Equation

CIES: Control, Instrumentation, and Electrical Systems

Chapter 1

1 Introduction

Simulators used in the field of nuclear power generation can be classified into physical simulators and software simulators. They can both be employed as platforms to support instrumentation and control (I&C) research on advanced control and verification and validation (V&V) for control designs on the multiple processes of nuclear power plants (NPPs). A software simulator, such as a training simulator or an engineering simulator, is usually developed in parallel with design and construction of a nuclear power plant.

To conduct I&C research on the thermal-hydraulic processes of NPPs, a physical simulator facility for a two-loop NPP (PWR or CANDU) is constructed in the Control, Instrumentation, and Electrical Systems (CIES) Research Laboratory at The University of Western Ontario. To extend the research, a software simulator for this facility is then developed in this thesis. The results and processes for the modeling, construction, and validation of the simulator will be presented.

For convenience, the term “Simulator” is used for “Software Simulator” in the thesis, unless otherwise stated.

1.1 Background on simulators

Simulation is a set of techniques, methods, and tools for developing a simulation model of a system and using the simulation model to study the behaviour of the system [1]. If the simulation is combined with enough powerful presentation tools, it can be considered as a simulator [2]. A simulator may have different purposes, such as assistance for designers or training of operators or as an aid for forming operation instructions.

Simulation tools can be traced back to the 1800s as mechanical devices constructed to demonstrate the mathematical relationships of physical quantities such as positions, angles, pressures, voltages, and so on [3]. Afterwards, simulation technologies and applications were developed with the advancements of science and technology. The application of simulation to NPPs, however, just emerged fifty-eight years ago. The advance of electronics makes it possible to produce a vast family of tools for the simulation of complex problems related to physics and engineering [2].

When an analog simulator was designed at the UK Atomic Energy Research Establishment of Harwell for training operators in 1957, it was constructed just with a control desk and certain instruments. From the 1960s to the 1970s, with digital technique coupled little by little, many nuclear power plant simulator training centers were in operation in the UK, the USA, France, Sweden, Germany, Spain, Japan, China (Taiwan), and Canada [2].

Computational simulation techniques for implementing mathematical models were also developed along with the development of high-speed computers. Especially after the accident in the Three Mile Island (TMI) nuclear power station and the issuing of the national standard on nuclear power plant simulators by the American National Standard Institute in 1979, nuclear power plant simulators were improved in their capability and performance. The recognition of the importance of software simulators was strengthened, not only with respect to training to deal with normal and accident conditions, but also for the study of specific industrial processes and advanced controls.

For the purpose of simulating the nuclear power process, besides training simulators, there is a distinct family of engineering simulators used by engineers rather than by trainees [2]. Training simulators can be classified into basic-principle simulators and full-scope simulators. Engineering simulators are considerably different, as they are used to aid engineers to understand the behaviours of plant processes under transient or accident conditions [2]. Accordingly, models used for different types of simulators are different. Training simulators run at the same speed as physical phenomena and processes and are also called real-time simulations. On the other hand, other simulators may not have such a requirement.

Each simulator has a reference unit or a simulation object, whether that is a whole plant or a specific system. The scope of the simulation depends on its purpose. A full-scope simulator considers details of the whole plant and can reproduce near-identical behaviour to that of the real plant. Comparatively, a part-task simulator or an engineering simulator may focus on a certain part of the plant with simplifications of other parts, while more details are ignored or considered as boundary conditions.

Since simulation technologies have been widely used in the development of nuclear power plant simulators for more than fifty years, there are a lot of vendors around the world that provide large-scale and full-scope nuclear training simulator platforms, such as GSE [11], WSC [12], L-3 MAPPS [13], CORYS [14], APROS [15], and TRAX [16]. These vendors have accumulated abundant experience in developing nuclear power plant simulators, and they are further applying new techniques in this field. For thermal-hydraulic systems of NPPs, extensive research activities are also conducted with a distinct family of software codes, such as RELAP5, COBRA, TRAC, ATHLET, and CATHARE. These codes and their coupling codes are used mainly for the analysis of thermal-hydraulic systems and neutronic transient responses as well as for the behaviours of NPPs under normal and accident conditions.

The simulator of a process is used not only for an industrial process like an NPP but also in the laboratory. Simulators work with mathematical models. With the development of high-speed computers, the simulators can choose more numerical methods as their solvers. Different types of mathematical models under different simplifications and assumptions are also implemented. In this research, the simulation technique is applied in the development of the simulator for a nuclear process test facility. The development process of this simulator will be presented herein.

1.2 Brief description of NPCTF

I&C systems are the backbone to support safe, reliable, and efficient operation of NPPs. To support research on I&C systems, a two-loop NPP simulator, known as Nuclear Process Control Test Facility (NPCTF), has been constructed in the Control, Instrumentation, and Electrical Systems Research Laboratory at The University of Western Ontario. The main objective of this simulator is to produce a replica of the true dynamic relationships in real NPP processes as seen by I&C systems [6]. To this end, some key devices of thermal-hydraulic systems of NPPs, such as a reactor, steam generator, pressurizer, and turbine, are taken into consideration in the design and construction of NPCTF.

In NPCTF, an electric heater (Heater) is employed to mimic the fission energy production in a reactor, and a chiller (Chiller) is used to simulate the process of heat exchange in a steam generator. The water level disturbed by vapor bubbles in a steam generator is simulated by a water tank called the heat exchanger behind tank (HX-Tank). A pressurizer tank (Pressurizer) represents the pressurizer and a turbine device (Turbine) is used to demonstrate the turbine in NPPs. They are both simplified based on their basic functions. To support multiple research areas, auxiliary components, that is, pumps, valves, pipes, and water-storage tanks, are incorporated into the facility to make sure the entire system can work effectively. Specially, thirty actuators and twenty-five sensors as well as the interface panels to connect industry-grade remote control systems are built in the facility [6].

A conceptual scheme of thermal-hydraulic components in this facility is depicted in Figure 1-1. The primary coolant loop is composed of the components including a main pump (Pump1), water valves, the Heater, the Chiller, and the Pressurizer. Energy conversion and heat exchange occur within this loop. The secondary water loop consists of the Chiller, a water pump (Pump4), a three-way valve, and a water cooler (Water Cooler). The purpose of this loop is to carry out the heat generated from the primary side in the Chiller to the Water Cooler, where the heat is then released to the environment. The primary loop and secondary loop are separated by the Chiller, which represents the thermal duty of the steam generator in NPPs. In addition to the two mentioned loops, there is another loop that works with the pressurized air. This pressurized air loop is composed of air valves, the Pressurizer, the HX-Tank, and the Turbine. In the HX-Tank, the air bubbles show the disturbance of the vapor in a steam generator. Similarly, in the Pressurizer, the pressurized air can create the effects of vapor on both inner pressure and water level. In the Turbine, furthermore, the basic relationship of the flow rate and the shaft speed is exhibited.

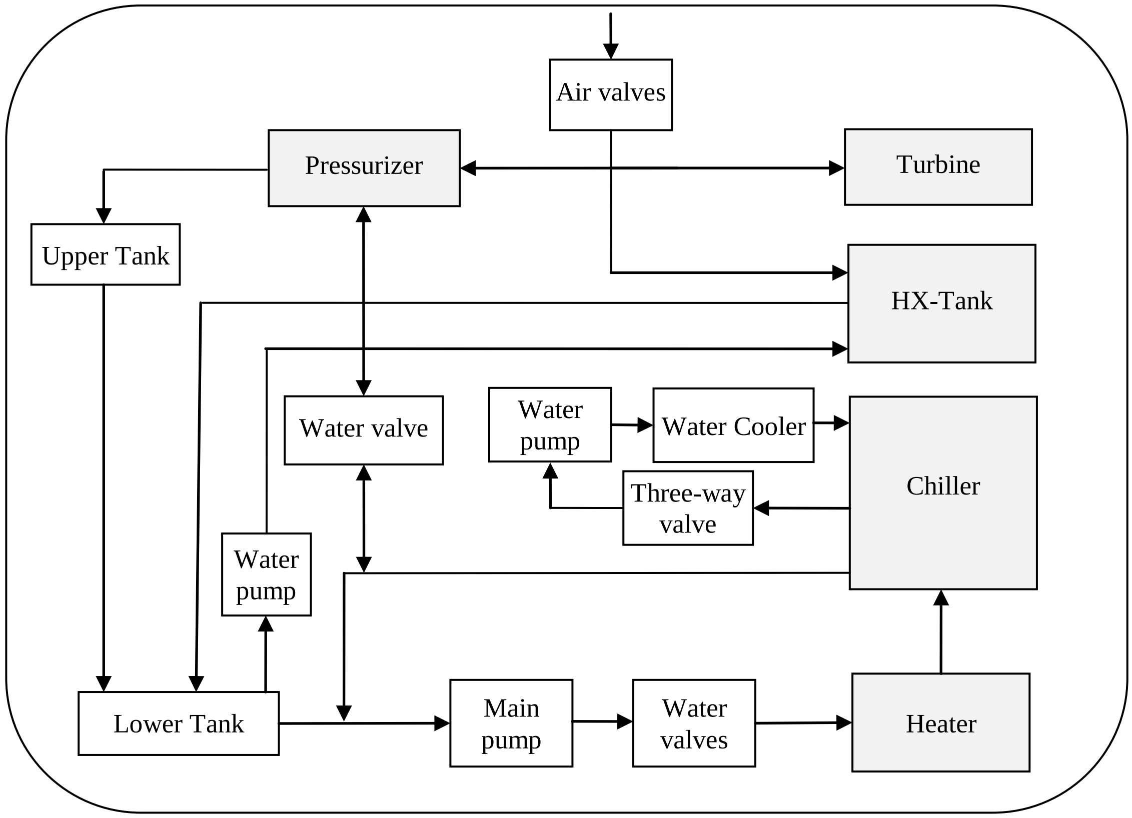


Figure 1-1 Conceptual scheme of thermal-hydraulic systems of NPCTF

According to the conceptual scheme, the thermal-hydraulic systems of NPCTF are constructed. Being designed as an engineering physical simulator, NPCTF has the ability to simulate the multiple dynamic behaviours of the thermal-hydraulic components found in an NPP. As the design purpose of NPCTF is has considered, several I&C devices as well as a powerful industry-grade control system from ABB are installed in order to monitor and operate the facility effectively in a real-time environment [6].

A front view of NPCTF is shown in Figure 1-2. Some major components are labelled therein. Those labelled Heater, Chiller, Pressurizer, HX-Tank, and Turbine will be discussed herein.

The piping and instrumentation diagram (P&ID) of NPCTF is shown in Figure 1-3. The denotations of the components and the sensors are cited herein.

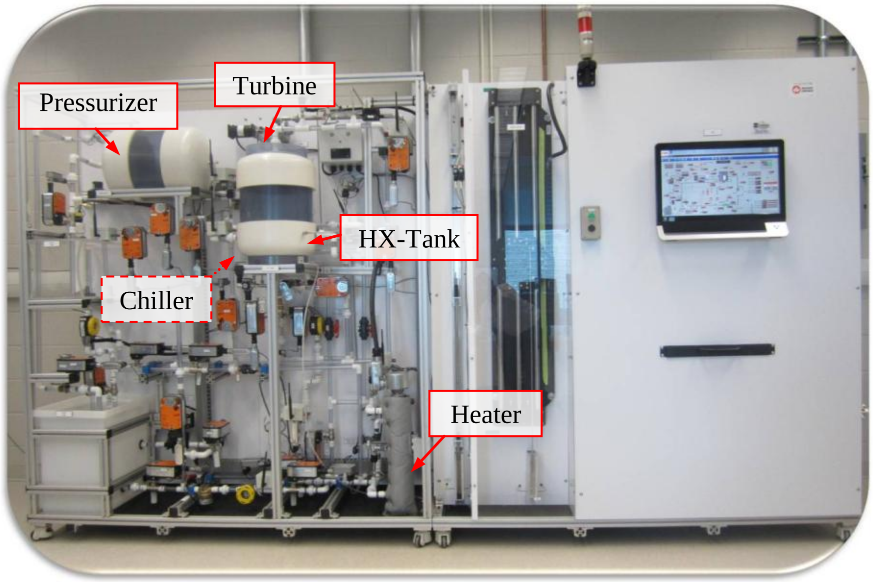


Figure 1-2 Physical implementation of NPCTF

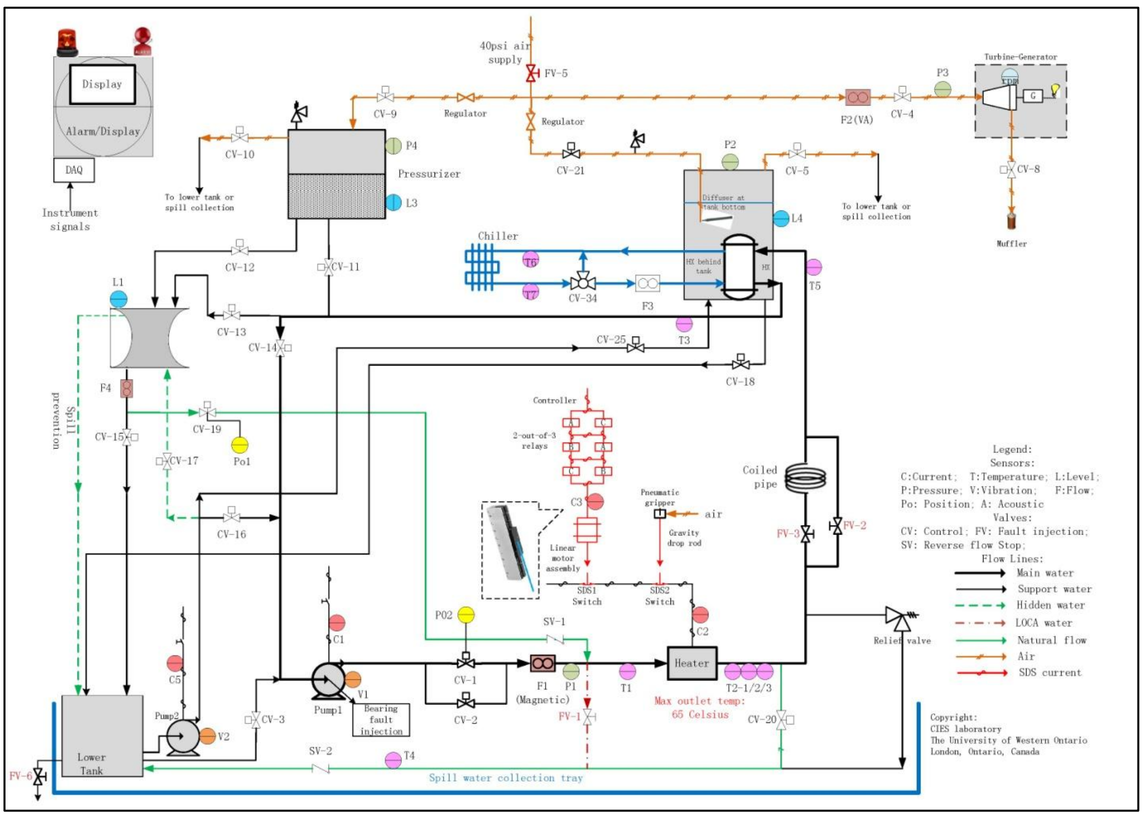


Figure 1-3 P&ID of NPCTF

1.3 Motivations and objectives

As a physical simulator, NPCTF can provide a real-time running environment where there are no timing issues in connecting to I&C systems. From another perspective, however, there are some drawbacks, as follows.

1) Due to the real-time environment, NPCTF cannot be accelerated or decelerated to test the dynamic response in a prolonged or a shortened duration. The response of a control design is inconvenient to be tested in NPCTF.
2) The running scenarios of NPCTF are observed by the sampling data from sensors. The sampling size and sampling time are limited by the data acquisition system in NPCTF.
3) There are limitations when studying or optimizing parameters of the processes in NPCTF, since the parameters of the facility cannot be easily changed. It is difficult to examine “extreme” operating conditions due to physical limitations.
4) Only one experiment can be conducted at one time on the facility. Time-sharing has to be made to accommodate multiple users.
5) It is difficult and sometime unsafe to operate the simulator outside the normal operating range to study the extreme conditions of the system, as experiments must be conducted within safe operating ranges of the process variables under the laboratory regulations.

To overcome the drawbacks of NPCTF and engage in further research into nuclear power generation processes, a software simulator regarding NPCTF as the simulation object is indispensable. As introduced in the preceding section, software simulators are widely used in the training and engineering of nuclear power plant processes. Software simulation can be used to validate behaviours of real systems. It is more cost-effective, less dangerous, and quick to test a control design.

Therefore, the purpose of this research is to develop a software simulator to simulate the dynamic characteristics of the thermal-hydraulic systems of NPCTF. The objectives of the development include:

- The development of the mathematical models for the dynamic processes of the thermal-hydraulic components;
- The development of the entire system models of the simulator in a simulation environment; and
- The tests and validations for the dynamic models as well as the entire simulator.

1.4 Modeling solution

Under the purpose of the research, the simulator of NPCTF should have these features:

- The models will be built by using physical principles based on appropriate simplifications and assumptions for the components.
- It focuses the scope on the certain part of NPCTF that is the thermal-hydraulic systems.
- The behaviours of the dynamic processes are the focuses to be tested and validated.

To implement the development of the simulator, firstly, the specifications of the simulator should be declared. Then the configuration of the simulator will be defined. By selecting a simulation environment, modeling and construction works can be continued until software simulations are executed. The simulator will be finally accepted under validations against the real behaviours of NPCTF.

1.5 Specifications

The American National Standards Institute (ANSI) has published a national standard for NPP simulators for use in operator training and examination [4]. It establishes the functional requirements for full-scope NPP control room simulators for use in operator training and examination, and establishes the criteria for the scope of simulation, the performance, and functional capabilities of simulators [4]. It is referenced for the simulator development in this research.

The objective of this research is to develop a software simulator. The specifications of the simulator are defined as follows.

- **Simulation object**

The facility with physical components, NPCTF, is the simulation object. This simulator will simulate the dynamic behaviours of NPCTF by developing and solving dynamic models of processes in NPCTF.

- **Simulation scope**

Through references to Figure 1-1, the conceptual scheme, and Figure 1-3, the P&ID of NPCTF, the scope of the simulator is illustrated in Table 1-1.

Table 1-1 Scope of the simulator

Category	Content
Main Components	Heater, Chiller, Pressurizer, HX-Tank, Turbine; Upper Tank, Lower Tank; Pump1, Pump2, Pump4; Water Cooler
Valves	CV-1, CV-2, CV-3, FV-2, FV-3, CV-11, CV-12, CV-14, CV-18, CV-25, CV-34; CV-4, CV-8, CV-9, CV-10, CV-5, CV-21, Regulators
Sensors	F1, F2, F3, P1, P2, P3, P4, T1, T2, T3, T5, T6, T7; L3, L4
Loops	Primary coolant loop, Secondary water loop, Pressurized air loop

- **Simulation functions**

Specifically, the simulation functions are included.

1) All the loops and components listed in Table 1-1 can work in normal conditions.
2) All the sensors listed in Table 1-1 can export data with correct units.
3) Parameters of components can be set with correct units.
4) Control valves can be adjusted to change flow rate in their working scope.
5) Water can be fed and drained into and out of the HX-Tank.
6) The Heater can be turned on and turned off, and the input current can be changed to control the input power.
7) Heat generated by the Heater can be transported with the fluid in the primary

coolant loop and can be released in the secondary water loop.

8) Pressurized air can adjust the inner pressures of the Pressurizer and the HX-Tank.

9) Pressurized air can drive the Turbine running along its speed curve.

10) Simulation data of dynamic processes can be obtained to validate simulation behaviours.

- **Modeling principles**

  All simulation models should be based on scientific principles. The balances of energy, momentum, and mass should be fulfilled.

- **Parameters**

  Parameters in the simulator should be consistent with NPCTF. Only those parameters that can be identified can be modified to improve simulation accuracy.

- **Simulation accuracy**

  Simulation accuracy should be assessed by sufficient validations between simulation results of the simulator and experimental results on NPCTF. Normally, static errors should be under 2%, and dynamic errors should be under 10%. Larger errors should be analyzed.

- **Acceptance**

  The simulator is finally accepted through V&V.

The scope of this research is limited within the specifications of the simulator. Other issues such as the running of NPCTF or the supporting simulation platform are not included in this research.

1.6 Contributions

Contributions of this research can be summarized as follows.

1) Construction of a software simulator for a physical simulator of an NPP, in this case NPCTF, to further research the dynamic processes of NPPs.

2) Development of mathematical models for dynamic processes in NPCTF to investigate the behaviours of thermal-hydraulic systems of an NPP by mathematical tools.

3) Implementation of the mathematical models into computer codes that can be executed continuously to simulate dynamic behaviours.

4) Test and identification of appropriate parameter settings of NPCTF based on the software simulator and NPCTF.

5) Conducting of V&V processes for the simulator to confirm the simulation models can indeed produce similar results, as those from the NPCTF.

1.7 Organization of the thesis

The rest of this thesis is organized as follows:

- The configuration of the simulator, the suitable modeling technique, and the dynamic models are introduced in Chapter 2.
- The Simscape scheme of the simulator is presented in Chapter 3.
- The creation of custom-defined blocks and the construction of the simulator diagram are described in Chapter 4.
- The V&V processes are presented in Chapter 5.
- The conclusion follows in Chapter 6, along with a summary of contributions and suggestions for future work.

Chapter 2

2 Configuration and Models for the Simulator

2.1 Configuration

2.1.1 Conceptual framework

According to the specifications of the simulator described in Chapter 1, NPCTF is the simulation object. The framework of the simulator is shown in Figure 2-1. The loop divisions and component connections are consistent with those in NPCTF.

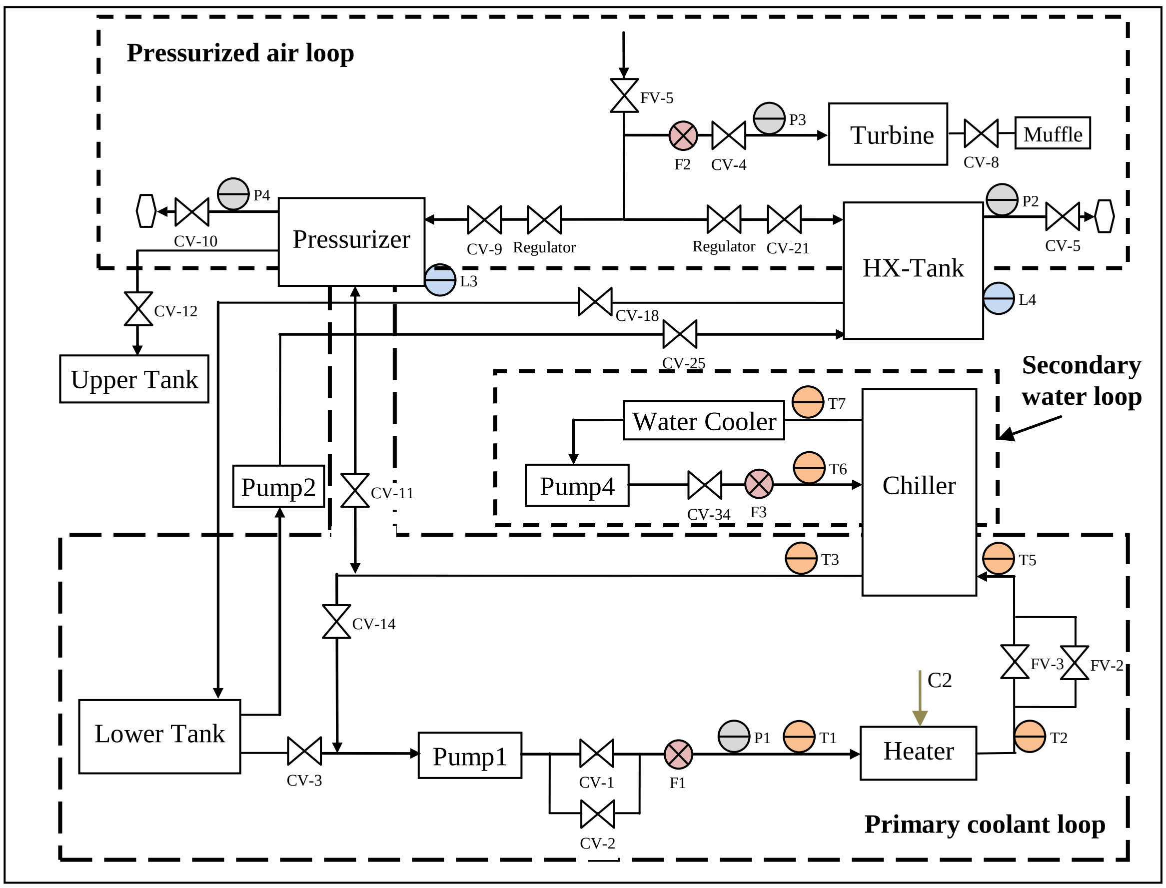


Figure 2-1 Framework of the simulator

2.1.2 Loop divisions

A loop in NPCTF is composed of multiple components connected physically, allowing fluids to move within it. Owing to the types of fluids, water and air, the loops of NPCTF can be divided into a hydraulic loop and a pneumatic loop. The simulator will also be constructed in line with the different loops.

A simulation loop is composed of boundaries (source and trap) and paths (branches) for fluids passing. Only integrated with complete boundaries and branches, the formed mathematical relation matrix can be solved uniquely. For hydraulic or pneumatic loops, a source or a trap should provide a constant pressure, and resistances in the branches should be considered, and other properties of fluids can also be presented in the source.

In general, the simulation system consists of the primary coolant loop, the secondary water loop, the pressurized air loop, and filling and draining water loops. Table 2-1 describes the components including valves and the sensors distributed in different loops.

Table 2-1 Loops and components in the simulator

Loops	Categories	Relevant Components
Primary coolant loop	Branches	Heater, Chiller, Pump1; CV-1, CV-2, CV-3, FV-2, FV-3, CV-11, CV-14
	Boundaries	Lower Tank, Pressurizer
	Sensors	F1, P1, T1, T2, T3, T5
Secondary water loop	Branches	Chiller, Pump4; CV-34
	Boundaries	Water Cooler
	Sensors	F3, T6, T7
Filling and draining water loops	Branches	Pump2, CV-18, CV-25, CV-12
	Boundaries	Pressurizer, HX-Tank, Upper Tank, Lower Tank
	Sensors	N/A
Pressurized air loop	Branches	Turbine, Regulators, CV-4, CV-8, CV-5, CV-21, CV-9, CV-10
	Boundaries	Pressurizer, HX-Tank
	Sensors	F2, P2, P3, P4

2.1.3 Loop connections

The primary coolant loop, secondary water loop, and pressurized air loop are the main loops in NPCTF. Filling and draining water loops constructed with fewer components are used for the HX-Tank and the Pressurizer. The connections between the loops and the main components forming dynamic processes are illustrated in Figure 2-2. There the dashed lines indicate the connected components as branches in the connected loops, and the solid lines indicate the connected components as the boundaries. Therefore, these connections will determine the roles of these components within a different loop.

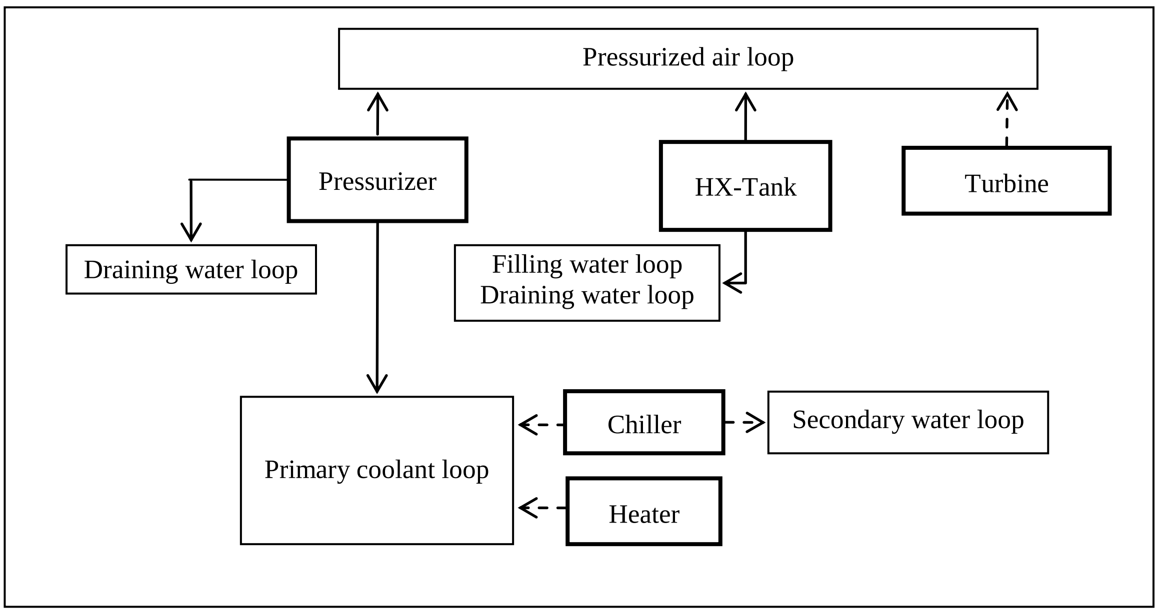


Figure 2-2 Loop connections

2.2 Modeling technique

2.2.1 Introduction

A model is an abstract representation of a real-life system. The most general categories of models are physical model, graphical model, and mathematical model [1]. The process of developing a model is known as modeling. A mathematical model represents the behaviours of a real system and the relationships of components mathematically. A good mathematical model can reproduce the behaviour of a system with little discrepancy. Also, more importantly, the model must be numerically stable. The numerical stability of a mathematical model is a prerequisite for any simulators.

There are two basic approaches for model construction: based on physical principles and via system identification. Modeling based on physical principles, or modeling based on first principles, is related to the knowledge of physics. By using this technique, a system is first broken down into subsystems whose properties and behaviours are known from the laws of mass, momentum, and energy. In this situation, it is essential to understand and formulize underlying relationships of real processes. Considering an electrical heater for water as an example, one can obtain the outlet temperature when the inlet temperature, the flow rate of the water, and the electrical power are known. The calculation proceeds through the energy conversion from electrical energy to thermal energy.

System identification is a process of constructing models based on measured input and output data through experiments. The method builds a mathematical model of a dynamic system by identifying the relationships of the specific input and output variables. In this situation, the relationships are determined just by fitting models to the measured data from experiments. The focus is only on the specific variables, neglecting other relationships. In many situations, system identification is a convenient modeling technique without getting to know the details of the underlying principles governing the processes. The determination of the model accuracy, however, is subject to the measurement errors of experiments.

Since the working principles of NPCTF are well understood, the modeling technique based on physical principles is selected to develop underlying mathematical models.

2.2.2 Modeling phase

A mathematical model is completed within three phases: 1) define problems; 2) formulate mathematical relations; and 3) express models in term of equations [3]. The three phases are explained below.

1) In the phase of problem definition, these items shall be clarified: a) purpose of the model; b) inputs, outputs, internal variables, and parameters of the model; and c) how the variables and parameters interact with each other. In this phase, assumptions and

simplifications are made. Due to the complexity of the reality, it is difficult to capture exact relations among these variables and parameters in a real process. A number of assumptions and simplifications have to be made to focus on the main relationships under the modeling purpose.

2) In the phase of formulating mathematical relations, conservation laws, state relationships, and constitutive equations are used. The conservation laws of mass, momentum, and energy depict the basic balances in a dynamic process. These basic principles applied to a specific process can be expressed in the form of differential equations:

a) The changed mass in a control volume per unit time = input mass flow rate – output mass flow rate.

b) The change of velocity is related to pressure and external momentum.

c) The change of power per unit time = power in – power out.

State relationships reflect the relationships between physical properties such as density, thermal conductivity, or specific heat. Constitutive equations describe the properties of certain materials or the relations of physical quantities such as the correlations of heat transfers. The correlation coefficients are normally derived from experiments and are valid in a narrow operating range.

3) In the third phase, an entire model is expressed following mathematical formulations. Due to the sequential executions of most programming languages, mathematical equations should be sorted in an appropriate order, especially if parameters calculated in a differential equation are to be used as the input variables to another equation.

In the modeling process, based on how much control volume a parameter represents, mathematical models can be classified into two types: a distributed parameter model and a lumped parameter model. In a lumped parameter model, space is reduced to a finite dimension, and partial differential equations of multidimensional space and time are simplified to ordinary differential equations with a finite number of parameters. A lumped parameter model can be transformed further to a set of first-order nonlinear differential equations. Thus, a differential-algebraic equation (DAE) set with no spatial variation can represent a lumped parameter model [9]. In this research, lumped parameter models are used.

2.3 Basic knowledge for thermal-hydraulic processes

Under the modeling technique based on physical principles, mathematical relations representing dynamic behaviours of thermal-hydraulic systems will be derived based on the relevant knowledge of general physics, heat transfer, thermodynamics, fluid dynamics, algebra, and geometry. They are briefly introduced in the following sections.

2.3.1 Fluid

Fluids in thermal-hydraulic systems include liquid and gas. Each fluid can be characterized by mass, density, volume, temperature, pressure, and velocity, while thermal properties are depicted by viscosity, thermal capacity, and thermal conductivity. In these parameters, pressure is calculated based on physics knowledge.

1) Pressure in liquid

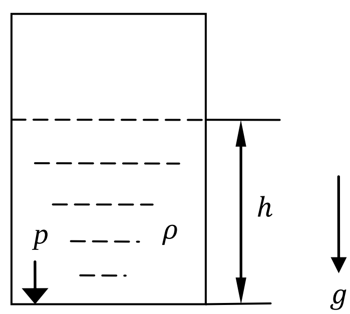


Figure 2-3 Pressure of a fluid column

The pressure of a liquid column depends on depth h and density ρ, as shown in Figure 2-3. If the liquid, such as water, is considered incompressible, the pressure of the liquid column is determined by the formula:

$$p = \rho g h \tag{2-1}$$

where g is gravity acceleration. In practice, once the density is determined, the pressure can be calculated with Equation 2-1.

2) Pressure of gas

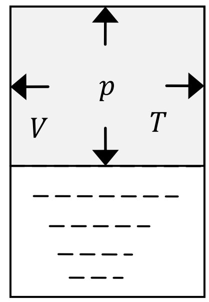


Figure 2-4 Pressure of gas in a closed space

Ideal gas is a theoretical gas under the assumption that molecules have no volume and do not interact. The ideal gas model can be used to approximate real gas under the conditions of lower density and relatively high temperature. According to the ideal gas law, the absolute pressure of the gas can be calculated by the formula:

$$p = \frac{nRT}{V} \tag{2-2}$$

where T, V denote thermodynamic temperature (K) and volume (m^3), respectively, as shown in Figure 2-4, n is the amount of substance, and the ideal gas constant $R = 8.314472\, J/(mol \cdot K)$ [10]. This formula can be specified for a specific gas by the specific gas constant, $R_{specific}$:

$$p = \frac{nRT}{V} = \frac{\frac{m}{M}RT}{V} = \frac{m}{V}\frac{R}{M}T = \rho R_{specific}T \tag{2-3}$$

where M is molar mass. The average molar mass of dry air, for example, is $28.97\ g/mol$. Hence, the specific gas constant of dry air is:

$$R_{air} = \frac{R}{M_{air}} = \frac{8.314472\ J/(mol \cdot K)}{28.97\ g/mol} = 0.287\ J/(g.K) \ = 287\ J/(kg.K) \tag{2-4}$$

In summary, the difference between the pressure calculations of a liquid and a gas is only in the dependent parameters. Under a status where the dependent parameters are available, the pressure can be determined by the above equations. In transient processes, as varying parameters, the depth of a liquid and the density of a gas should be obtained first by transient calculations.

2.3.2 Heat transfer

There are a number of fundamental concepts related to the process of heat transfer.

1) Convection heat transfer

Often referred to simply as convection, this is the transfer of heat from one place to another by combining the processes of conduction and advection. It is common that the convection heat transfer occurs between a moving fluid and a boundary surface of different temperatures, as illustrated in Figure 2-5.

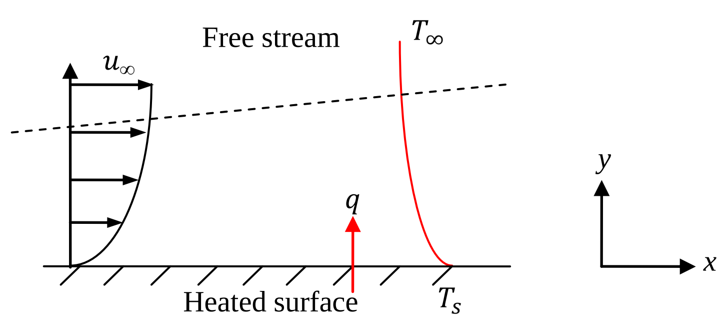


Figure 2-5 Convection heat transfer

The appropriate rate equation is of the form:

$$q = h(T_s - T_\infty) \qquad (2\text{-}5)$$

This expression is known as Newton's law of cooling [8], where q, as the convective heat flux (W/m^2), is proportional to the convective heat transfer coefficient h ($W/(m^2 \cdot K)$), and proportional to the difference between the surface temperature T_s and the fluid temperature T_∞. The convective heat transfer coefficient h depends on the conditions in the boundary layer, in which it is influenced by surface geometry, the nature of the fluid motion, and the assortment of fluid thermodynamic and transport properties. A more practical approach to calculating h is from the empirical relations of appropriate dimensionless groups. Next, the overall heat transfer rate is determined by the product of the heat flux and the area of heat transfer, that is:

$$Q = q \cdot A = hA(T_s - T_\infty) \tag{2-6}$$

2) Convection boundary layers

The concept of boundary layers is central to the understanding of convection heat transfer. Between a solid surface and the fluid flowing over it, there is a velocity boundary layer and a thermal boundary layer.

The region of the flow development from the leading edge of a plate or a tube in which the effects of viscosity are observed is the velocity boundary layer. Since viscous forces act on the fluid near to the surface, the velocities of fluid, denoted as m/s, form a gradient at the surface, as illustrated in Figure 2-5. The dynamic viscosity, denoted as $\mu\ (kg/(m \cdot s))$, is used to demonstrate the viscous force in an incompressible and isotropic Newtonian fluid. The viscous force is described by the shear stress τ_s, which is expressed simply as:

$$\tau_s = \mu \frac{\partial u}{\partial y} \tag{2-7}$$

Just as the velocity boundary layer develops over a surface where there is a velocity gradient of fluid flow, a thermal boundary layer is defined as the region where temperature gradients exist. The heat flux at the surface can be obtained by applying Fourier’s law [8]:

$$q = -k \frac{\partial T}{\partial y}|_{y=0} \tag{2-8}$$

where k is the thermal conductivity of a fluid ($W/(m \cdot K)$).

3) Laminar flow and turbulent flow

The first step to deal with any convection problem is to determine the flow conditions, that is, whether the boundary layer is laminar or turbulent. Laminar flow is a streamlined flow occurring when the fluid is flowing in parallel layers with no disruptions between the layers. Turbulent flow is a flow regime characterized by chaotic property changes. Usually the low velocities result in laminar flow. Turbulent flows are always irregular.

4) Dimensionless groups

The dimensionless quantities, which are the Reynolds number, Prandtl number, and Nusselt number, are usually used for convection heat transfer.

The Reynolds number is computed by:

$$Re = \frac{\rho u_{\infty} x}{\mu} \tag{2-9}$$

where u_{∞} is the velocity of the free stream (m/s), x is the distance from the leading edge as the characteristic length, and ρ, μ represent the density (kg/m^3) and dynamic viscosity of fluid ($kg/(m \cdot s)$), respectively. The Reynolds number is a ratio of the inertia to the viscous forces of fluids. It can describe whether the flow conditions lead to laminar or turbulent flow. The location of the transition from laminar to turbulent is determined by the critical Reynolds number.

The Prandtl number is also dimensionless when a consistent set of units is used:

$$Pr = \frac{\nu}{\alpha} \tag{2-10}$$

where ν, α represent kinematic viscosity (m^2/s) and thermal diffusivity (m^2/s), respectively, which can be computed by $\nu = \frac{\mu}{\rho}$, $\alpha = \frac{k}{\rho Cp}$. The kinematic viscosity conveys information about the rate at which the momentum diffuses through the fluid. The thermal diffusivity describes the diffusion of heat in the fluid. Therefore, the Prandtl number is defined as a ratio of the momentum diffusivity to the thermal diffusivity. It provides a measure of the relative effectiveness of momentum and energy transport by diffusion in the velocity and thermal boundary layers [8].

The Nusselt number is defined as:

$$Nu = \frac{hL}{k} \quad \text{(2-11)}$$

where L is the characteristic length, and h, k, respectively, denote the heat transfer coefficient and the thermal conductivity of a fluid. The Nusselt number is a ratio of convection to pure conduction heat transfer across the boundaries. It may be obtained by analytical solution or by empirical correlations. When considering all the fluids, the algebraic expression, as below, is assumed:

$$Nu = CRe^{m}Pr^{n} \quad \text{(2-12)}$$

where C, m, n are constants determined from experimental data. Empirical correlations for a wide range of geometries are available in the literature [8]. Although there are a lot of empirical correlations suitable for most engineering calculations, in practice, the exact values for the convection coefficients in specific situations are rarely provided. This is due to uncertainties such as the flow turbulence or the surface roughness. Thus, the empirical correlation coefficients may be modified to fit specific experimental data.

Once the Nusselt number is obtained by the empirical correlations, the convective heat transfer coefficient h can be calculated by Equation 2-11, and then the heat transfer rate can be computed by Equation 2-5 and Equation 2-6.

5) Radiation

Thermal radiation is energy emitted by matter with a certain temperature and is expressed as:

$$E = \varepsilon\sigma T_s^4 \quad \text{(2-13)}$$

where T_s is the absolute temperature (K) of the surface, σ is the Stefan-Boltzmann constant ($\sigma = 5.67 \times 10^{-8}\ W/(m^2 \cdot K^4)$), and ε represents the emissivity of a surface ($0 \leq \varepsilon \leq 1$).

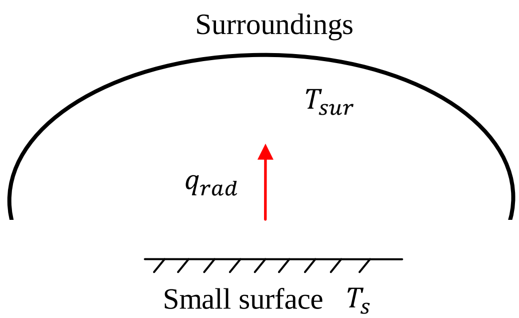


Figure 2-6: Radiation heat transfer

In a case where the radiation exchange occurs between a small surface at T_s and a much larger and isothermal surface at T_{sur} that completely surrounds the smaller one, as shown in Figure 2-6, the net rate of radiation heat transfer per unit area q_{rad} is calculated by:

$$q_{rad} = \varepsilon\sigma(T_s^4 - T_{sur}^4) \quad (2\text{-}14)$$

2.3.3 Transient process

In a transient process, the basic balance equations can be obtained by applying conservation laws, which are given in the following form [17].

1) Mass balance

A differential equation of mass conservation can be depicted as:

$$\frac{\partial\rho}{\partial t} + \frac{1}{A}\frac{\partial(A\rho v_z)}{\partial z} = S \quad (2\text{-}15)$$

where S is a source or sink of mass that represents the flow into or out of a boundary, A is the area, and v_z is the velocity in the direction of the fluid flow.

2) Momentum balance

The momentum equation describes the change of the momentum in a control volume. It equals the momentum flow rate into the control volume minus the momentum flow rate out of the control volume plus the net external force caused by pressure, gravity, friction, and inertial force. It can be shown as:

$$\frac{\partial(\rho v_z)}{\partial t} + \frac{1}{A}\frac{\partial(A\rho {v_z}^2)}{\partial z} = -\frac{\partial p}{\partial z} - \rho g sin\vartheta + \tau_s \frac{P_f}{A} + \rho f_z \qquad (2\text{-}16)$$

where ϑ is the angle of the direction (for vertical case $sin\vartheta = 1$), τ_s is the shear stress, P_f is the wetted perimeter, and f_z denotes inertial force. In the case where the shear stress and inertial force are neglected, that is, $\tau_s \frac{P_f}{A} + \rho f_z = 0$, the change in velocity is then subject to the pressure and the gravity.

3) Energy balance

By applying conservation of energy for a moving fluid under transient conditions, an equation of energy balance can be obtained in the form:

$$\frac{\partial(\rho e)}{\partial t} + \frac{1}{A}\frac{\partial(A\rho e v_z)}{\partial z} = q''' - \frac{P_h}{A}q'' - \rho g v_z sin\vartheta + \tau_s \frac{P_f}{A} v_z + \rho f_z v_z \qquad (2\text{-}17)$$

where e is the stagnation internal energy per unit mass, which can be expressed as:

$$e = u_e + \frac{1}{2}{v_z}^2 \qquad (2\text{-}18)$$

and u_e is the internal energy per unit mass, q''' is a volumetric heat source, q'' is the wall heat source or heat sink, and P_h represents the heated perimeter. It is shown that the change in the internal energy is related to the generated internal heat, the sum of heat in and out, and the additional thermal energy added from potential energy, viscous forces, and external forces.

2.3.4 Discussion

The knowledge described herein forms the basis to develop dynamic models for NPCTF. More specially, the pressure equations are used in the models of the Pressurizer and the HX-Tank. The equations related to heat transfer processes are used in the models of the Heater and the Chiller. These two types of equations are used under steady-state conditions. For dynamic relationships, the principles of the transient process are applied based on the balance equations.

In the transient process, the mass balance is involved in the models of the Pressurizer and the HX-Tank, which are connected to the water loop and the air loop. If water is assumed to be incompressible, in Equation 2-15, the time item $\frac{\partial \rho}{\partial t} = 0$. Otherwise, for compressible air, since $\frac{\partial \rho}{\partial t} \neq 0$, air density or air mass should be expressed with a differential equation.

With regard to the momentum balance in Equation 2-16, usually the shear stress and inertial force are neglected ($\tau_s \frac{P_f}{A} + \rho f_z = 0$), and the velocity can be expressed as mass flow rate $A\rho v_z = F$, and if the effect of altitude difference is neglected ($sin\vartheta = 0$), the momentum balance can then be simplified as a relationship of the pressure and the flow rate.

For the energy balance expressed in Equation 2-17, by neglecting the interface dissipation energy contributed by the viscous force and external force ($\tau_s \frac{P_f}{A} v_z + \rho f_z v_z = 0$), as well as the conversion from kinetic energy ($\frac{\partial(\frac{1}{2}\rho {v_z}^2)}{\partial t} = 0$) and potential energy ($\rho g v_z sin\vartheta = 0$), the change of internal energy for incompressible water ($q''' = 0$) is then related simply to the wall heat source or heat sink. This is used to model heat exchanges in the Heater and the Chiller.

2.4 Dynamic models

To construct the simulator of NPCTF, mathematical models for dynamic processes are the foundation to build the simulator system. With the modeling technique based on physical principles and the relevant physical knowledge, dynamic mathematical models for the Heater, Chiller, Pressurizer, HX-Tank, and Turbine are developed. The main mathematical relations of these models are explained in the following sections.

2.4.1 Heater

In the Heater, electrical energy is converted into thermal energy to mimic the energy conversion in a reactor. The generated heat is carried out by the coolant, in this case, water, to the Chiller. As a signal to represent the result of the dynamic process, the outlet temperature is the main output variable. The input current and voltage, the inlet temperature, and the flow rate of fluid are the main input variables.

To simulate the processes of heat generation and heat transfer to the working fluid, some assumptions and simplifications made to the Heater model are defined as follows.

1) The input power of electricity is assumed to be lumped at one point, where it is converted to thermal energy and then is transferred to the metal sheath of the electrical heating rod. The heat transfer between the metal sheath and the working fluid water is also considered at one point with uniform temperatures. Thus, lumped parameter models are employed.

2) The interface dissipation energy contributed by shear force, kinetic energy, and potential energy are ignored, since they are very small. The volumetric heat source is also ignored, because the expansion work of water is much too small in comparison to the energy converted from electricity.

3) The external force caused by gravity, friction, and inertial force to momentum are neglected. Momentum balance is then simplified as a relationship of pressure and flow rate.

4) The properties of water, such as density, thermal conductivity, specific heat, and dynamic viscosity, are assumed to be just temperature-dependent under the relatively low pressure and low temperature (<100 ℃).

As a result, the model of the Heater is composed mainly of three ordinary differential equations (ODEs) and three algebraic equations. The three ODEs are used to capture the dynamic relations of the process.

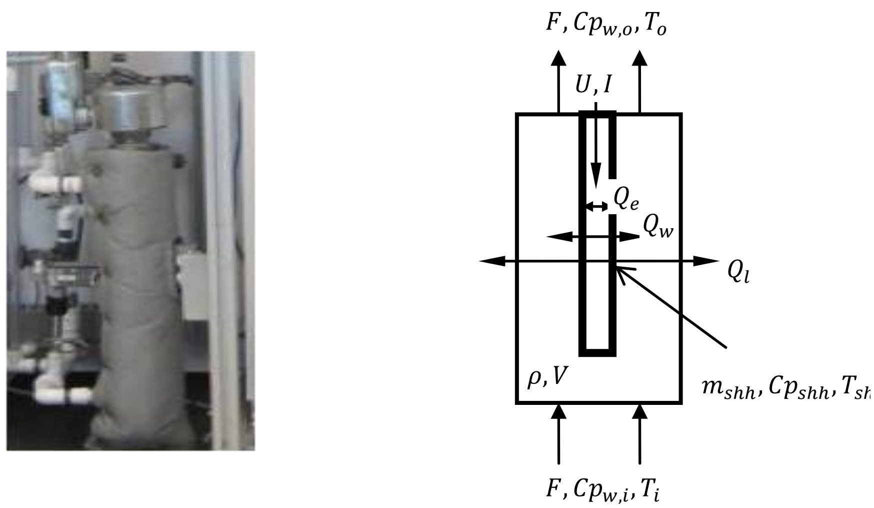


Figure 2-7 Illustration of the Heater model

1) The process of the power generated and converted to thermal energy is represented as:

$$\tau \frac{dQ_e}{dt} = \eta UI - Q_e \qquad (2\text{-}19)$$

where the generated heat $Q_e = \eta P_e = \eta UI$, η denotes the efficiency of the energy conversion, and U, I are voltage and current, respectively; the time constant τ reflects the time required for the heat to transfer to the entire metal wall of the electrical heating rod.

2) The transient thermal balance on the water side is expressed by:

$$\frac{d(\rho_w V Cp_{w,o} T_o)}{dt} = F Cp_{w,i} T_i + Q_w - F Cp_{w,o} T_o - Q_l \qquad (2\text{-}20)$$

The heat exchange between the water and the metal sheath of the electrical heating rod is expressed by the algebraic equation $Q_w = hA(T_m - T_o)$, where the metal temperature, T_m, is the temperature of the metal sheath of the electrical heating rod, T_{shh}. Convective heat transfer coefficient h determines the situation of heat transfer on the solid surface, whose calculation will be explained in the Chiller model.

The heat lost to the ambience is described as the algebraic equation $Q_l = \varepsilon\sigma A({T_o}^4 - {T_{amb}}^4)$, where ε, σ denote the coefficients of the heat lost to the ambience and the Stefan-

Boltzmann constant. The temperature of the outer wall of the Heater is supposed to be the same as the outlet temperature of the heated water, T_o. T_{amb} denotes the ambient temperature.

Specific heat at the inlet and outlet, $Cp_{w,i}$ and $Cp_{w,o}$, is determined by the temperatures at the inlet and the outlet, respectively. In practice, they are obtained by looking up tables.

The flow rate F at the inlet and outlet are the same value under the assumption of momentum.

In a transient status, water density ρ_w and specific heat $Cp_{w,o}$ are constants that are determined by water temperatures. Therefore, Equation 2-20 can be rewritten as:

$$\rho_w V Cp_{w,o} \frac{dT_o}{dt} = F Cp_{w,i} T_i + Q_w - F Cp_{w,o} T_o - Q_l \tag{2-20a}$$

3) The thermal balance of the metal sheath of the electrical heating rod is described as:

$$\frac{d(m_{shh} Cp_{shh} T_{shh})}{dt} = Q_e - Q_w \tag{2-21}$$

where the thermal conduction of the metal material incoloy alloy is considered to be effective with a constant specific heat Cp_{shh}; m_{shh} is the equivalent mass of the metal sheath. Since m_{shh}, Cp_{shh} are constants, Equation 2-21 can be rewritten as:

$$m_{shh} Cp_{shh} \frac{dT_{shh}}{dt} = Q_e - Q_w \tag{2-21a}$$

This is the transient equation of the metal temperature T_{shh}. Since $Q_w = hA(T_{shh} - T_o)$, this equation is a typical first-order ODE as a lumped parameter model.

2.4.2 Chiller

There are two paths in the Chiller: the primary side and the secondary side. They link to the primary coolant loop and the secondary water loop, respectively. The duty of the Chiller is to transfer the heat of the primary side to the secondary side through a series of connected metal lamellas. Similar to the Heater, the outlet temperatures are the main variables to describe the dynamic processes in the Chiller.

The model of the Chiller is separated into a primary side and a secondary side by the metal lamellas. Lumped parameter models are applied to equations of heat exchange processes between water and metal. Similar assumptions and simplifications to those of the Heater are made of the Chiller. Then three differential equations are used to describe the characteristics of the dynamic processes in the Chiller.

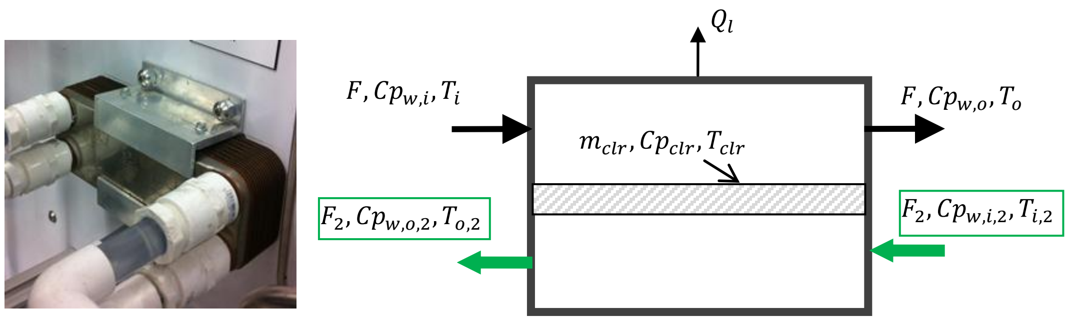


Figure 2-8 Illustration of the Chiller model

1) The transient thermal balance of the primary side is described as:

$$\frac{d(\rho_w V_1 Cp_{w,o} T_o)}{dt} = FCp_{w,i}T_i - FCp_{w,o}T_o - Q_1 \qquad \text{(2-22)}$$

where the volume of the primary side V_1 is a constant. Similar to the Heater model, Equation 2-22 is rewritten as:

$$\rho_w V_1 Cp_{w,o} \frac{dT_o}{dt} = FCp_{w,i}T_i - FCp_{w,o}T_o - Q_1 \qquad \text{(2-22a)}$$

2) On the secondary side, the transient thermal balance is expressed as:

$$\frac{d(\rho_{w,2} V_2 Cp_{w,o,2} T_{o,2})}{dt} = F_2 Cp_{w,i,2}T_{i,2} - F_2 Cp_{w,o,2}T_{o,2} - Q_2 \qquad \text{(2-23)}$$

where the subscript *2* denotes the secondary side. Similarly, this equation can be rewritten as:

$$\rho_{w,2} V_2 Cp_{w,o,2} \frac{dT_{o,2}}{dt} = F_2 Cp_{w,i,2}T_{i,2} - F_2 Cp_{w,o,2}T_{o,2} - Q_2 \qquad \text{(2-23a)}$$

3) For the metal lamellas, the thermal balance is described with the change of thermal capacity:

$$\frac{d((m_{clr}Cp_{clr})T_{clr})}{dt} = Q_1 - Q_2 - Q_l \qquad (2\text{-}24)$$

As a typical ODE, since the equivalent mass of the metal lamellas m_{clr} and the specific heat Cp_{clr} are considered unchanged, Equation 2-24 can be rewritten as:

$$m_{clr}Cp_{clr}\frac{dT_{clr}}{dt} = Q_1 - Q_2 - Q_l \qquad (2\text{-}24a)$$

As a lumped parameter model, the metal lamellas are considered as a point with mass and identical properties. In the practice of the simulation, the equivalent mass of the metal lamellas m_{clr} is identified with an appropriate value to reflect the dynamic process of the metal.

Prior to each transient equation, heat transfer rates Q_1, Q_2, Q_l are calculated by algebraic equations with expressions like Equation 2-6 and Equation 2-14.

In the thermal processes in the Heater and the Chiller, the convective heat transfer coefficient h depends on the complex conditions of the solid surface and the fluid. In this research, it is calculated from the Nusselt number. From the definition of the Nusselt number in Equation 2-11, it is calculated by $h = \frac{kNu}{D}$.

The empirical correlations of the Nusselt number, as shown in Equation 2-12 ($Nu = CRe^m Pr^n$), provide the approach to obtain the value of the Nusselt number. By referring the correlation coefficient of external flow through a flat plate [8] and by revisions in the model validations, the empirical correlations of the Nusselt number, the Reynolds number, and the Prandtl number are finally described as:

$$Nu(Re) = \begin{cases} Re \le Re_l,\ Nu_l = 0.664Re^{0.50}Pr^{0.33} \\ Re \ge Re_t,\ Nu_t = 1.56Re^{0.85}Pr^{0.33}\ (heater),\ \ Nu_t = 0.316Re^{0.8}Pr^{0.33}(chiller) \\ Re_l < Re < Re_t,\ Nu_l + (Nu_t - Nu_l)\frac{Re - Re_l}{Re_t - Re_l} \end{cases} \qquad (2\text{-}25)$$

where the subscripts *l* and *t* represent the margins of laminar flow and turbulent flow, respectively.

2.4.3 Pressurizer

The duty of the Pressurizer is to stabilize the pressure of the primary coolant loop. In NPCTF, the Pressurizer is simplified as a pressurized horizontal cylinder linking to the air loop and the water loop. The elements in the Pressurizer are only water and air. They represent the subcooled water and superheated vapor of an industrial pressurizer. Thus, instead of heating steam, compressed air is used to control the inner pressure, and the inner pressure is turned into the main output variable.

In view of the modeling purpose, some assumptions and simplifications for the model of the Pressurizer are defined as follows.

1) The water in the Pressurizer is homogenous and uncompressible. This makes the relationship between the mass flow rate and the volumetric flow rate of water fixed by water density.

2) The air is compressible and is treated as dry air. The air is considered as an ideal gas satisfying the ideal gas law.

3) The Pressurizer is sealed well, and there are no other air leakages except the spraying path.

4) Heat transfers in the Pressurizer, to and from the Pressurizer wall, are very small in magnitude and then are negligible. The process in the Pressurizer is adiabatic. Only the hydraulic effect is considered.

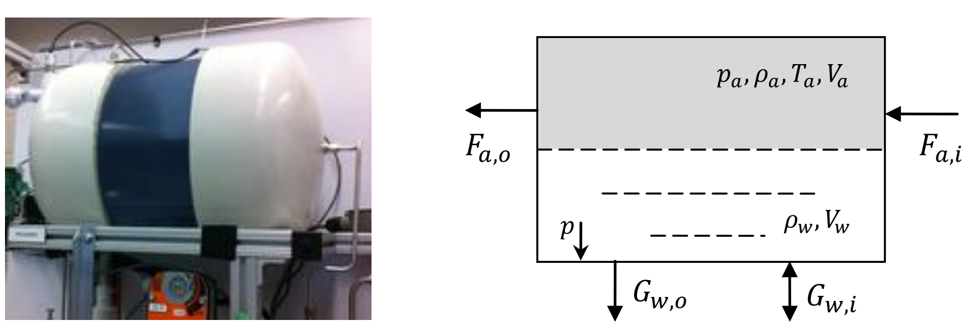


Figure 2-9 Illustration of the Pressurizer model

The change of the water volume in the Pressurizer is due to the difference in the flow rate of the entered water and the flooded water. It is represented simply by:

$$\frac{dV_w}{dt} = G_{w,i} - G_{w,o} \tag{2-26}$$

where $G_{w,i}$ is the flow rate of the valve connecting to the primary coolant loop. $G_{w,i}$ may be positive or negative, for maintaining the pressure balance in both sides of the connecting valve. $G_{w,o}$ is the volumetric flow rate of water draining to the Upper Tank.

In order to keep the pressure of the primary coolant loop in a stable condition, the inner pressure in the Pressurizer is regulated through compressed air. According to conservation of mass, the mass change of the air is determined by the amount of air entered into and released from the Pressurizer, that is,

$$\frac{dm_a}{dt} = \frac{d(\rho_a V_a)}{dt} = \Sigma F_{a,i} - \Sigma F_{a,o} \tag{2-27}$$

The air absolute pressure is calculated by the state function of the ideal gas. It is expressed as:

$$p_a = \rho_a R_a T_a = \frac{m_a}{V_a} R_a T_a \tag{2-28}$$

where the specific gas constant of dry air $R_a = 287\ J/(kg.K)$, as Equation 2-4 shows.

The pressure at the bottom of the Pressurizer is then obtained by the water pressure plus the air pressure, which is:

$$p = p_w + p_a = \rho_w g L_w + p_a \tag{2-29}$$

where L_w is the water level calculated from the volume of the stored water.

The relationship of the water and the air is that the sum of their volume is equal to the volume of the tank, which is expressed as:

$$V_a + V_w = V = \frac{\pi D^2}{4} l \tag{2-30}$$

where l is the length the cylinder of the Pressurizer, and D is the diameter of the Pressurizer tank.

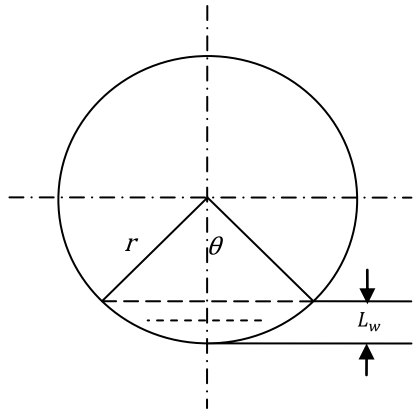


Figure 2-10 Illustration of the cross of the Pressurizer cylinder

By neglecting the bulges of both sides of the Pressurizer horizontal cylinder, the approximation of the water level is calculated by using the first order of Taylor expansion. It is described as:

$$L_w = \begin{cases} L_w < r, V_w < \frac{V}{2},\ L_w = r - rcos\frac{\theta}{2} = r - rcos\frac{\left(6\times\frac{V_w}{\frac{1}{2}r^2\cdot l}\right)^{\frac{1}{3}}}{2} \\ L_w > r, V_a < \frac{V}{2},\quad L_w = r + rcos\frac{\theta}{2} = r + rcos\frac{\left(6\times\frac{V_a}{\frac{1}{2}r^2\cdot l}\right)^{\frac{1}{3}}}{2} \end{cases} \qquad (2\text{-}31)$$

where the radius of the cylinder $r = \frac{D}{2}$.

2.4.4 HX-Tank

The HX-Tank is used to demonstrate the hydraulic process of a steam generator with the change of water level. In NPCTF, the HX-Tank is a vertical cylinder. It is linked to the water loop and the air loop as their boundaries. When the pressurized air enters the tank at the bottom, the water stored in the tank is disturbed by the floating air bubbles, and then the produced air pressure in the upper space of the tank can depress the amount of bubbles.

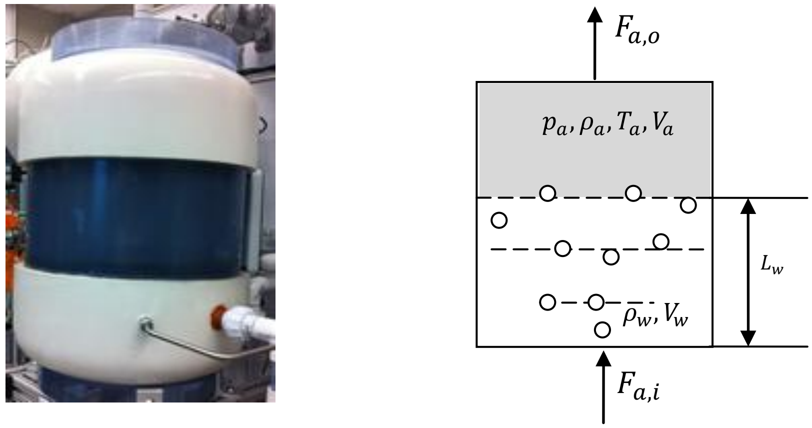


Figure 2-11 Illustration of the HX-Tank model

Based on the same assumptions and simplifications as the Pressurizer, the computations of the water pressure and the air pressure are similar to those of the Pressurizer. Considering the influence of air bubbles, the water level of the HX-Tank is demonstrated by:

$$L_W = \frac{V_W + \xi \frac{\Sigma F_{a,i}}{\rho_a{}^4}}{\frac{\pi D^2}{4}} \qquad (2\text{-}32)$$

where ξ is used as a bubble coefficient, and the bulges of both edges of the HX-Tank are neglected. This equation is only used to demonstrate the relationship of the water level and the influence of air bubbles without canonical derivation.

2.4.5 Turbine

In NPCTF, the duty of the Turbine is simplified to demonstrate the relationship between the air flow and the shaft speed. The shaft speed is the main output variable.

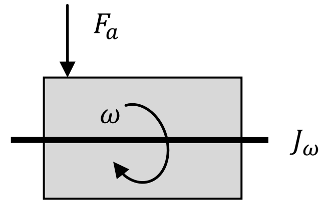


Figure 2-12 Illustration of the Turbine model

From Newton's Law, the differential equation for a shaft rotating at the speed ω is defined as:

$$J_\omega \frac{d\omega}{dt} = \zeta \Sigma F_a - f\omega \tag{2-33}$$

where ζ denotes the conversion rate from the kinetic momentum of air flow, J_ω represents the moment of inertia of the Turbine shaft, and $f\omega$ represents the output power of the Turbine with the friction factor f.

2.4.6 Summary of the models

All the mathematical relations are expressed through differential equations and algebraic equations. They are derived from conservation laws, state relationships, and constitutive equations. Differential equations, which are mostly ordinary differential equations (ODEs), represent the calculations of the time-dependent variables in dynamic processes. Algebraic equations, on the other hand, describe the relations at steady state. The variables in algebraic equations also vary, following those time-dependent variables in differential equations. All the mathematical relations constitute the foundation to construct a simulation system.

2.5 Summary

To fulfill the specifications of the simulator, a configuration is conceived and described through the framework, loop divisions, and loop connections. The framework shows that the construction of the simulator system is on the base of the physical networks of

NPCTF. The simulated loops in the simulator capture the main loops of NPCTF under the simulation functions.

The modeling technique based on physical principles is used to develop mathematical models for dynamic processes in NPCTF. The relevant knowledge for thermal-hydraulic systems is introduced. Then, the dynamic models of the main components are presented.

Chapter 3

3 Simscape Scheme for the Simulator

3.1 Simulation environment

To achieve the objectives of this research based on the configuration of the simulator, the modeling technique based on physical principles is employed to develop the software models. As described in the preceding sections about the modeling techniques and the basic physical knowledge as well the dynamic models, the details of the underlying principles of the NPCTF system are investigated and prepared. To implement the construction of a simulation system and to execute the constructed simulator, a supporting simulation environment is needed.

Matlab Simulink is a powerful tool in research and simulation for plant processes. It provides a graphical modeling environment that includes expandable toolboxes and the interactive graphical editor [24]. Simscape, as a toolbox in Simulink, supports modeling and simulating physical systems through a physical network approach spanning hydraulic, thermal, pneumatic, and other physical domains. It also provides built-in fundamental block libraries covering lots of industrial processes and a tool to develop extended custom-defined block libraries. The embedded solution of modeling physical processes is suitable to build the simulator of NPCTF. The main features and useful information about Simscape in Simulink can be seen in Appendix A, or reference the official website of Mathworks (*http://www.mathworks.com*).

3.2 Simscape scheme

Based on Simscape, the diagram of a simulation system is constructed by assembling component blocks under their working physical domains. An underlying mathematical model represents the functionality in each component block. The integration of the component blocks in the diagram of the simulator system can form a working mathematical relation matrix of the original system. By employing a definite solver, the integrated mathematical relation matrix can be solved to present the simulation results.

The forming and solving of the mathematical matrix under the simulator system are the duty of the simulation platform, in this case, Simulink with the toolbox Simscape.

3.2.1 Source of component blocks

The relevant blocks mentioned in Table 2-1 above, should have their specific blocks. The component blocks can be from the built-in fundamental libraries of Simscape or the developed custom-defined block libraries.

Table 3-1 summarizes the source of the components in the simulator. The custom-defined blocks need to be developed. The built-in blocks can be directly used, either one or a combination of more than one. The last row of this table lists the built-in blocks that may be used in the simulator under the feature of Simscape.

Table 3-1 Source of components

Components	Source of Blocks	
	Custom-defined Blocks	Built-in Blocks
Heater, Chiller, Pressurizer, HX-Tank, Turbine	Heater, Chiller, Pressurizer, HX-Tank, Turbine	N/A
CV-1, CV-2, CV-3, FV-2, FV-3, CV-11, CV-14, CV-34, CV-12, CV-18, CV-25,	Water Valve	N/A
CV-4, CV-8, CV-5, CV-21, CV-9, CV-10, Regulators	Air Valve	N/A
Upper Tank, Lower Tank	N/A	Constant Volume Hydraulic Chamber
Pump1, Pump2, Pump4	N/A	Lookup Table, Product, Constant, Hydraulic Pressure Source
F1, F3	N/A	Hydraulic Flow Rate Sensor
F2	N/A	Pneumatic Mass & Heat Flow

		Sensor
P1	N/A	Ideal Hydraulic Pressure Sensor, Hydraulic Reference
P2, P3, P4	N/A	Pneumatic Pressure & Temperature Sensor
T1, T2, T3, T5, T6, T7	N/A	Ideal Temperature Sensor, Thermal Reference
Others	N/A	Solver Configuration, Custom Hydraulic Fluid, Gas Properties, Pneumatic Pressure Source Hydraulic Resistive Tube, Fluid Inertia, Pneumatic Atmosphere Reference

3.2.2 Construction guidelines

According to the requirements of simulation functions and the features of Simulink and the toolbox, Simscape, the key guidelines of constructing the simulator in Simscape are defined as follows.

1) The loops, including the primary coolant loop, the secondary water loop, the filling and draining water loops for the HX-Tank and Pressurizer, and the pressurized air loop, are constructed by connecting the component blocks from the built-in block library and the custom-defined block library. The employed built-in blocks will be used under their present functionalities. The custom-defined blocks will be developed under their mathematical models.

2) Besides the components listed in the configuration of the simulator as shown in Figure 2-1 in Chapter 2, more blocks, such as Resistive Tube or Pneumatic Atmosphere Reference, may be added into the loops to confirm their integrity.

3) Under the definition of a loop, blocks in loops are identified as branches or boundaries. A component block can work in more than one loop. The connections in different loops are achieved by the different physical ports embedded in blocks. The definitions of the ports determine the physical domains that the blocks work in. The details will be presented in the following sections.

4) The measurement sensors of flow meter, pressure, and temperature in the loops are placed at their correct positions, as shown in Figure 2-1. The measurement of the water level is exported from the component blocks. In addition, more sensor blocks can be placed under the needs of monitoring or collecting the simulation data.

5) Water tanks, including the Lower Tank and Upper Tank, are assumed to be full of water. All the water pipes are considered as working with the same geometrical shape factor, the same internal surface roughness height, and the same laminar and turbulent margins.

6) The hydraulic resistances of the components, including pipes, valves, and other branches in the hydraulic domain, are given with estimated values, according to their structures and positions. The resistances may be modified to more suitable values in validations. The air valves, regulators, and other pneumatic branches in the air loops are also given with estimated pneumatic resistances. The pneumatic resistances should be in the same magnitude in order to ensure that the loops work without issues.

7) Simulation loops in Simscape work in different physical domains. Under the definitions of built-in domains and the categories of the employed built-in blocks, the hydraulic, thermal, and pneumatic domains are involved in the modeling process. The hydraulic domain and the pneumatic domain do not transport heat along with fluids in the current version of Matlab. Hence, heat transport must be achieved in the thermal domain.

3.2.3 Working physical domains

Simscape divides the physical networks into different physical domains. As mentioned in the preceding section, the implementation of the simulator construction in Simscape faces the issue of how to use the thermal domain to deal with heat transportation. Generally, in this simulator, the heat transport just occurs among the Heater, Chiller, and Water Cooler, which corresponds to heat generation and heat diffusion. Thus, the thermal domain is applied along with the hydraulic domain in the primary coolant loop and the secondary water loop. The working physical domains that have blocks involved are indicated in Table 3-2, in which the "Source of Blocks" corresponds to Table 3-1 and the "Working Loops" corresponds to Table 2-1.

Table 3-2 Working physical domains of components

Components	Source of Blocks (Custom-defined(C) or Built-in(B))	Working Loops	Working Domains
Heater	Heater (C)	Branch of Primary coolant loop	Hydraulic, Thermal
Chiller	Chiller (C)	Branch of Primary coolant loop, Secondary water loop	Hydraulic, Thermal
Pressurizer	Pressurizer (C)	Boundary of Primary coolant loop, Pressurized air loop, and Filling water loop	Hydraulic, Pneumatic
HX-Tank	HX-Tank (C)	Boundary of Pressurized air loop, Filling water loop, and Draining water loop	Hydraulic, Pneumatic
Turbine	Turbine (C)	Branch of Pressurized air loop	Pneumatic
CV-1, CV-2, CV-3, FV-2, FV-3, CV-11, CV-14, CV-34, CV-12, CV-18, CV-25,	Water Valve (C)	Branch of Primary coolant loop, Secondary water loop, Filling water loop, and Draining water loop	Hydraulic
CV-4, CV-8, CV-5, CV-21, CV-9, CV-10, Regulators	Air Valve (C)	Boundary of Pressurized air loop	Pneumatic
Upper Tank, Lower Tank	Constant Volume Hydraulic Chamber (B)	Boundary of Primary coolant loop, Filling water loop, and Draining water loop	Hydraulic

Pump1, Pump2, Pump4	Lookup Table (B) Hydraulic Pressure Source (B)	Branches of Primary coolant loop, Secondary water loop, and Filling water loop	Hydraulic
F1, F3	Hydraulic Flow Rate Sensor (B)	Branches of Primary coolant loop, Secondary water loop	Hydraulic
F2	Pneumatic Mass & Heat Flow Sensor (B)	Branch of Pressurized air loop	Pneumatic
P1	Ideal Hydraulic (B) Pressure Sensor (B) Hydraulic Reference (B)	Attached in Primary coolant loop	Hydraulic
P2, P3, P4	Pneumatic Pressure & Temperature Sensor (B)	Attached in Pressurized air loop	Pneumatic
T1, T2, T3, T5, T6, T7	Ideal Temperature Sensor (B) Thermal Reference (B)	Attached in Primary coolant loop and Secondary water loop	Hydraulic

The three embedded domains defined in Simscape will also be used in the creations of the custom-defined component blocks with the Simscape language. Since some components work in more than one loop with different roles as branches or boundaries at the same time, their physical conserving ports linking to other components are defined corresponding to the working physical domains in the Simscape environment.

3.2.4 Branches and boundaries

Under the physical network approach in Simscape, the blocks are divided into branches and boundaries. Branches should provide the relationship of the pressure and the flow rate, and boundaries should provide pressures.

Fluid branches in this simulator may contain hydraulic resistance or pneumatic resistance. Based on the law of momentum, the relationship of the pressure difference Δp and the flow rate G can be expressed as:

$$\Delta p = R \cdot G \tag{3-1}$$

where R denotes hydraulic resistance or pneumatic resistance. The altitude difference in components is neglected. The relationship of Equation 3-1 may be used in the blocks of the Heater, Chiller, and Turbine, which are designed as hydraulic branches or pneumatic branches, as shown in Table 3-2.

3.3 Summary

A simulation environment is the condition under which to implement the configuration of the simulator. With the features to support physical modeling, the toolbox in Simulink of Matlab, Simscape, is selected as the simulation environment.

The Simscape scheme of the simulator is described in identifying the source of the model blocks, designing the simulator construction, and distinguishing the working physical domains of the component blocks. Next, the construction of the diagram of the simulator will be conducted by developing the custom-defined blocks and drawing the built-in blocks from Simscape libraries.

Chapter 4

4 Construction of the Simulator

4.1 Creation of custom-defined blocks

The component blocks, including the Heater, Chiller, Pressurizer, HX-Tank, Turbine, and Water Valve and Air Valve, are developed as custom-defined component blocks. The programming tool in Simscape, Simscape language, is used to carry out the development.

4.1.1 Simscape language

Simscape language is the tool to build custom-defined component blocks in Simscape. It is a text-based programming language with features for physical modeling. According to the specification of the Simscape language, ports, variables, parameters, and equations of the blocks are all defined in textual files. The ports include two types: conserving ports (circle symbol) and signal ports (triangle symbol). The diagram of the simulator is built by linking the ports of the blocks to form a physical network.

The common sections used in the block files are illustrated in Figure 4-1. The output variables (section of outputs) and internal variables (section of variables) are the results calculated by the equations with the parameters (section of parameters) and the input variables (section of inputs). For each block, the number of output variables and internal variables should be no more than the number of equations. This is to ensure the equation matrix can be solved. The output variables create output signal ports in the block. The internal variables can be valued in initialization, which is declared in the function setup section.

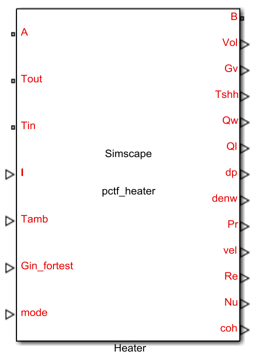
All physical quantities declared in the textual files are given with physical units permitted by Simscape. Unit conversions are carried out automatically by Simscape.

```
component pctf_heater < foundation.hydraulic.branch % This is an
example; it declares name, working physical domain (only for using
the built-in definitions in Simscape).

  nodes
  % declare physical ports by using the type of conserving port.
  end

  inputs
  % declare the input variables as signal ports.
  end

  outputs
  % declare the output variables as signal ports.
  end

  variables
  % declare the internal variables calculated in the block.
  end

  parameters
  % declare the constants which appear in the block dialogue box.
  end
  parameters(Access = private)
  % declare the parameters used in the internal calculations.
  end

  function setup
  % initialize variables and check errors.
  end

  branches
  % set up the relations of variables and ports, or define
association of through and across variables.
  end

  equations
  % the expressions of mathematical relations.
  end
end
```

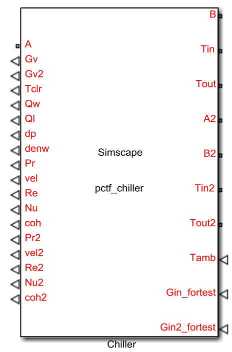
Figure 4-1 Declaration of block sections

4.1.2 Custom-defined component blocks

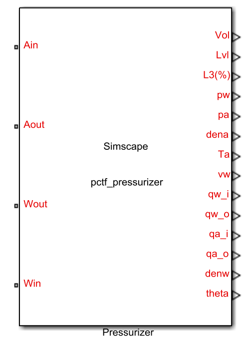
By using the Simscape language, the schemed custom component blocks are developed. The complete Simscape codes of the custom-defined blocks are presented in Appendix B. The generated blocks are shown as follows, in Figure 4-2.



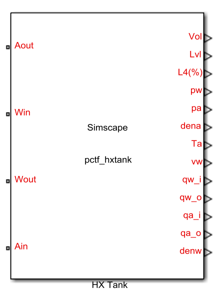
(1) **Heater**



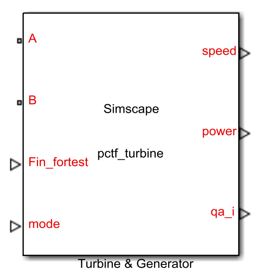
(2) Chiller



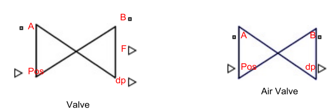
(3) Pressurizer



(4) HX-Tank



(5) Turbine



(6) Water Valve and Air Valve

Figure 4-2 Custom-defined blocks

1) Heater: The block of the Heater works in the hydraulic domain and the thermal domain. It contains a pair of conserving hydraulic ports and a pair of conserving thermal ports.

2) Chiller: Similar to the Heater, it is built with two hydraulic branch definitions containing two pairs of conserving hydraulic ports as well as two pairs of conserving thermal ports.

3) Pressurizer: The Pressurizer block is defined as the boundary of the hydraulic domain and the pneumatic domain. Four conserving ports in hydraulic and pneumatic are defined.

4) HX-Tank: This block is similar to the Pressurizer block. Four conserving ports in hydraulic and pneumatic are engaged.

5) Turbine: This block is designed as a pneumatic branch with two conserving pneumatic ports.

6) Valves: The valve blocks are separated as the water block and the air block. They are used in water loops and pneumatic loops. A pair of conserving ports is used to link other blocks.

4.1.3 Block ports

The figures of blocks show the two types of ports: physical conserving ports and physical signal ports.

1) Physical conserving ports (circle symbol) are used to connect to the ports of other blocks of the same type. In the textual files of blocks, the physical conserving ports are declared in the section of nodes. A physical conserving port can be considered as a package of parameters and variables defined in domain definitions.
2) Physical signal ports (triangle symbol) are used as unidirectional ports for transferring internal signals in a diagram. They are declared in the sections of inputs and outputs. The directions of the ports are indicated by the directions of triangle symbols.

All the denotations, descriptions, and units of the ports of the developed custom blocks are shown in Appendix C.

4.1.4 Variables and parameters

In the textual file declaring the blocks, as shown in Figure 4-1, the section of variables defines the internal variables used in the calculations of the section of equations, and the section of parameters defines the parameters used as independent constants. The values of variables declared in the textual files are initial values for calculations. The values and units declared in the section of parameters can be modified in the dialogue box of blocks.

All the parameters of blocks can be classified into three types:

a) Structure parameters such as length, width, height, diameter, area, and volume. These parameters use the actual values from NPCTF.

b) State and constitutive parameters, such as properties of water, gas or metal, resistance, correlation coefficients, the friction factor, the margins of laminar and turbulent flows, etc.

c) Initial parameters, which provide the values of the initial conditions for the differential equations. The suitable initial parameters can accelerate the convergence of the iteration process.

4.2 Construction of loops

The Simscape scheme of the simulator was designed in the preceding chapter. Under the construction guidelines of the simulator, along with the prepared component blocks from built-in libraries and the custom-defined block library, the diagram of the simulator is constructed. The Simscape guidelines of how to build simulation diagrams can be referenced in Appendix A.5 and the user manual from Mathworks [22], [23], [24].

4.2.1 Primary coolant loop

Figure 4-3 shows the key connections of the components in the primary coolant loop, in which the Heater, Chiller, Pump1, and water valves are the hydraulic branches to be linked at hydraulic ports. The Pressurizer is connected at the hydraulic port. Some pipes are modeled with the built-in block of Hydraulic Resistive Tube. A block of Solver Configuration is mandatory for a modeling diagram. For the topologically distinct hydraulic circuit, a Custom Hydraulic Fluid block is required which declares some global water parameters for the connecting loops.

Specifically, the thermal domain works between the Heater and the Chiller. As the Simscape scheme of the simulator discussed, the thermal connections are just for the purpose of transporting temperature signals. The inserted blocks of thermal inertia, named Thermal Heater and Thermal Chiller, are used to reflect the thermal delays in fluid moving processes.

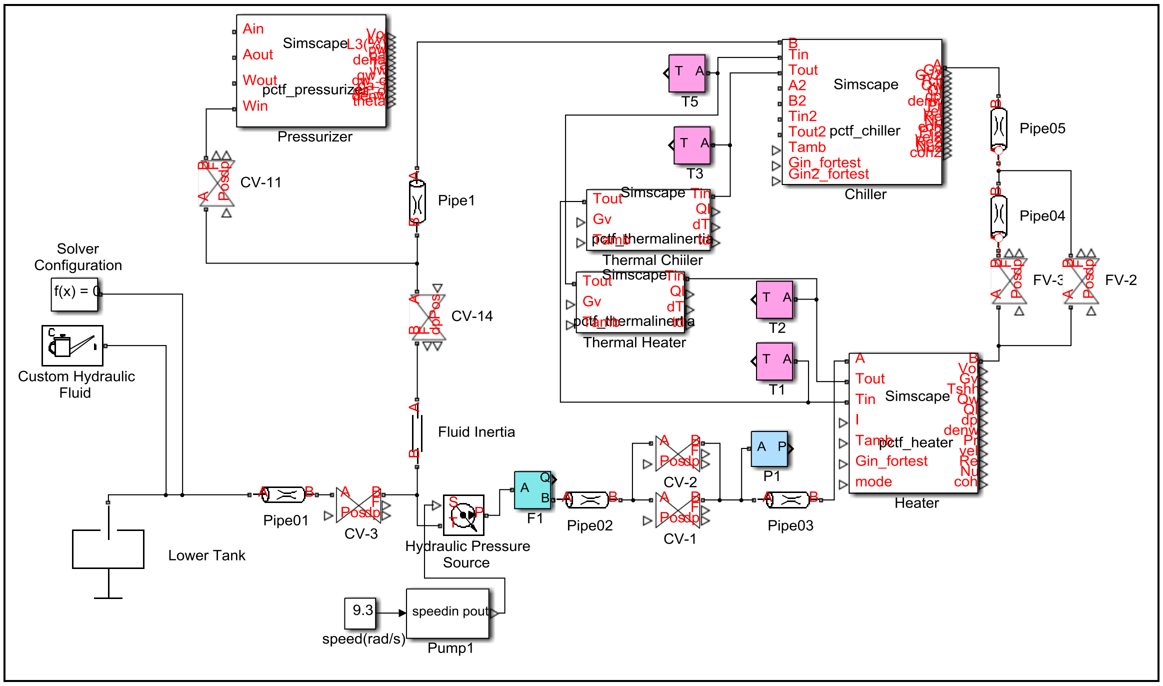


Figure 4-3 Diagram of the primary coolant loop

4.2.2 Secondary water loop

The secondary water loop is composed of the four components, the Chiller, the Pump4, the valve CV-34, and the Water Cooler, as Figure 4-4 shows. The blocks of the Chiller, Pump4, and CV-34 compose a hydraulic loop, while the Water Cooler is connected to the Chiller in the thermal domain to decrease temperatures along with fluid moving in the hydraulic loop. The block of Hydraulic Resistive Tube is used to demonstrate hydraulic resistances.

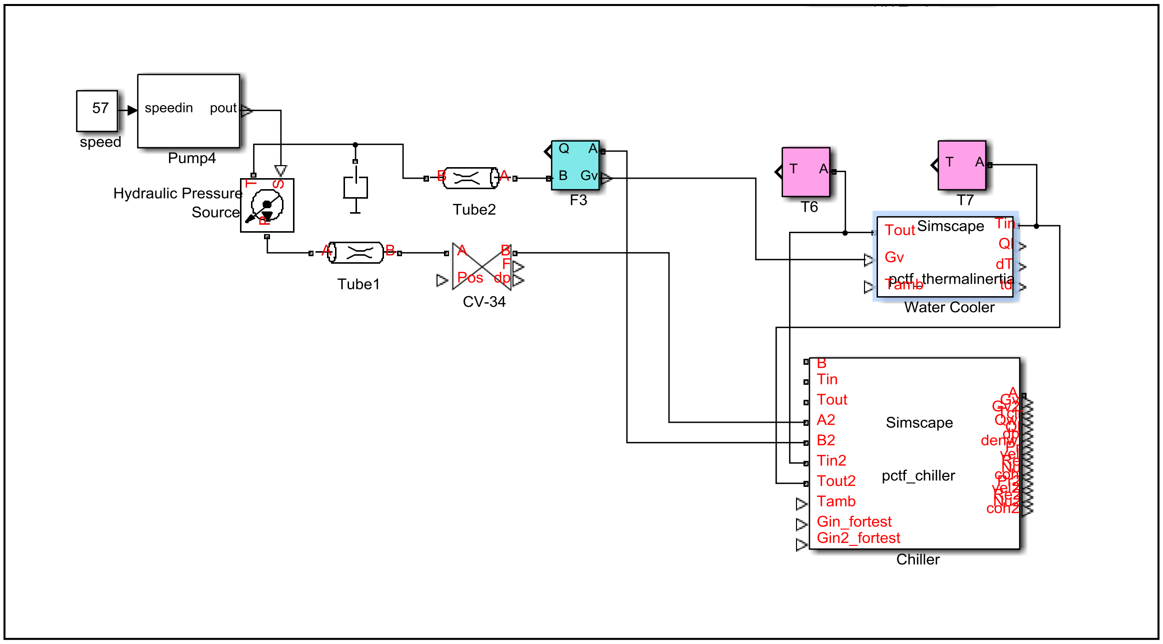


Figure 4-4 Diagram of the secondary water loop

4.2.3 Pressurized air loop

Figure 4-5 shows briefly the pressurized air loop. It consists of three subsystems for the Pressurizer, HX-Tank, and Turbine. The Pressurizer and the HX-Tank are the boundaries in this loop, while the Turbine is a pneumatic branch. As a solvable loop, four blocks of Pneumatic Atmospheric Reference are necessary as sources or traps of the loop, which represent the atmospheric environment. A built-in block of Pneumatic Pressure Source is added at the inlet to set the pressure of the compressed air. Similar to the block Custom Hydraulic Fluid in the hydraulic loop, the block Gas Properties provides some setting of air parameters for the air loop.

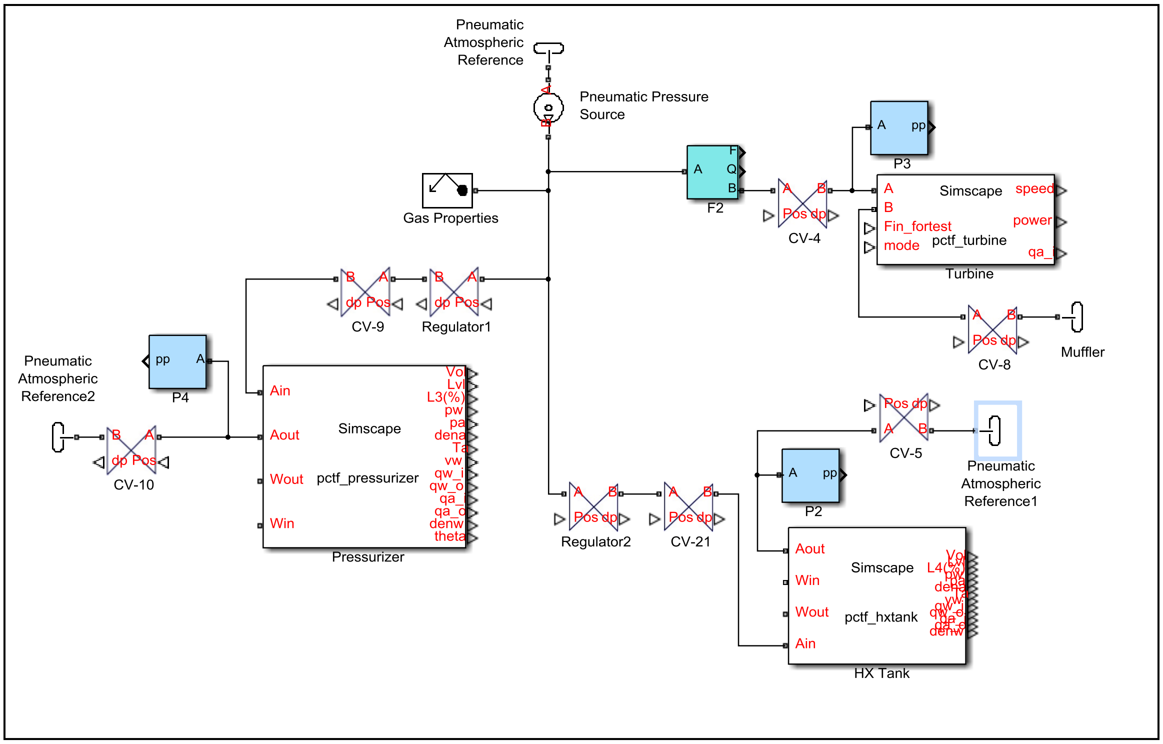


Figure 4-5 Diagram of the pressurized air loop

4.2.4 Loops of filling water and draining water

The loops for filling and draining water of the HX-Tank are mainly composed of the Lower Tank, HX-Tank, Pump2, and the valves CV-18 and CV-25. The Lower Tank is the hydraulic boundary. The Pump2 and the valves as well as additional hydraulic pipes are hydraulic branches.

The draining water loop of the Pressurizer is simple, with one valve CV-12 as the hydraulic branch. The Pressurizer and the Upper Tank play the roles of hydraulic boundaries.

Figure 4-6 shows the three actual hydraulic loops.

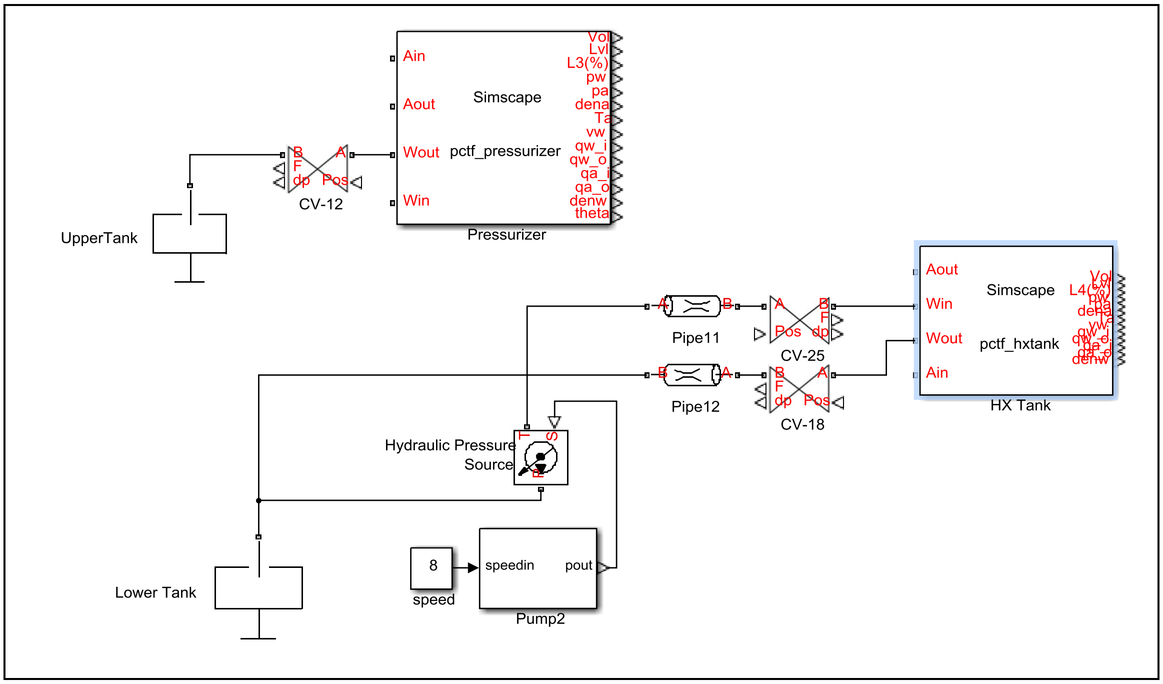


Figure 4-6 Diagram of the loops for filling and draining water

4.3 Monitoring of variables

In the diagram of the simulator, there are three ways to monitor the output values of blocks including sensors and component blocks.

a) Outputs of blocks. Variables that have been defined as outputs of the blocks can be connected to a Display block to show their values.

b) Scope block. One or more variables can be connected to a Scope block, where their trends are displayed.

c) Workspace. Workspace in Matlab is a memory space for storing and sharing variables [22]. When outputs of blocks are connected to a Workspace block, the variables are stored in the workspace as a data matrix. The data can be dealt with in Matlab or can be exported.

Figure 4-7 shows an example of monitoring the simulation variables. The blocks Display, Scope, and Workspace are drawn from the standard Simulink block library and are used to show unitless values. To convert Simscape signals with physical units to Simulink unitless signals, the block of PS-Simulink Converter (PS_S) is inserted between the blocks of Simscape and Simulink. The output signal units can be specified in this block, and unit conversions are handled automatically.

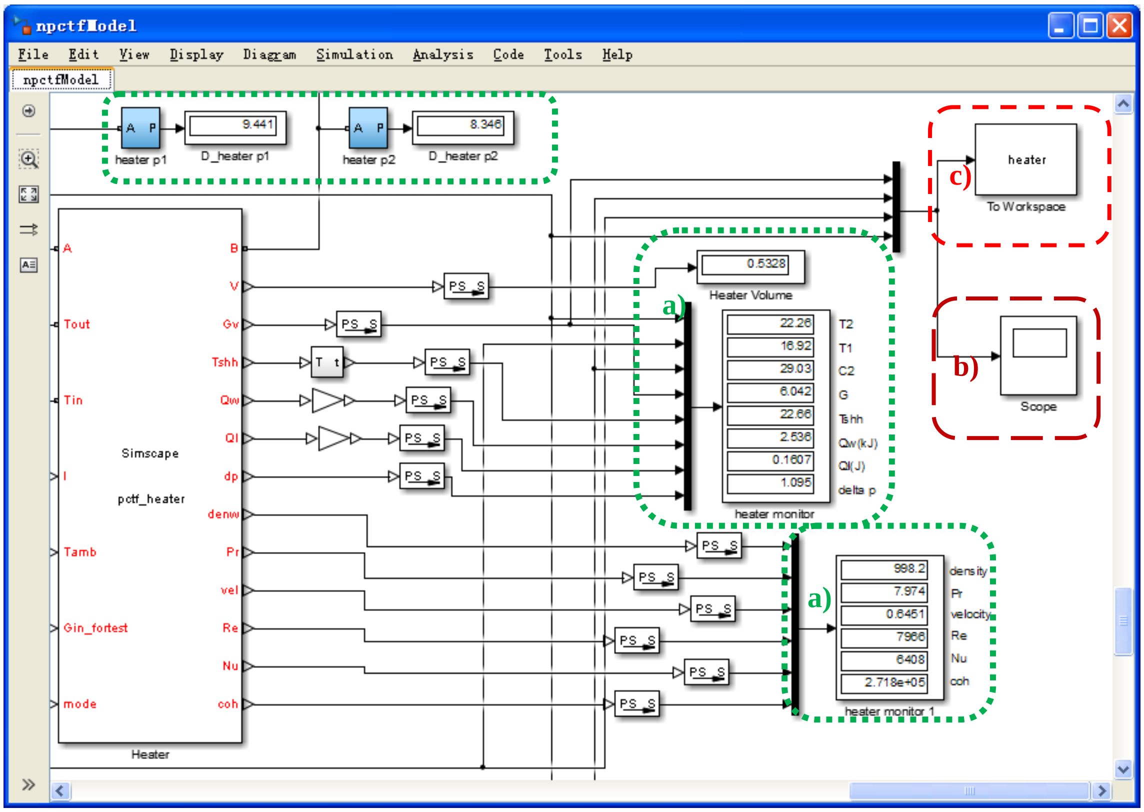


Figure 4-7 Monitoring simulation variables in the Heater

4.4 Global model configuration

In Simscape, as global model configuration parameters, the simulation time, step size, and solver type, are given before running the simulator system.

4.4.1 Simulation time and step size

Simulation time is the time consumed by simulation activity. It is usually not the same as actual clock time, since it can be paused, accelerated, or decelerated. Step size is the simulation time in which the entire model is executed once. A fixed-step size means that the simulation in each step consumes the same time.

In the development of the simulator, an implicit fixed-step solver is identified as a correct solver choice to execute this simulation system. Through tests, many step sizes can be used with numerical stability and model solvability. Herein, 0.125 second is used due to concerns of computer load and simulation speed near to the actual clock time.

4.4.2 Solver choice

A solver is the numerical method of solving the equation matrix of all the mathematical models. Hydraulic loops and pneumatic loops in this simulator diagram have different time scales. In Simscape, the unit of flow rate in hydraulic loops uses volumetric flow rate (m^3/s), while that in pneumatic loops uses mass flow rate (kg/s). Sometimes the pressurized air shows very small values in the air flow rate ($\sim 10^{-6}$), which may result in unstable issues. To preclude the unstable issues, an implicit solver is the suitable choice for this simulator diagram.

Compared to an explicit solver, an implicit solver provides greater stability for oscillatory behaviours, and generates a Jacobian matrix and solves the set of algebraic equations at every time step, using a Newton-like method [24]. The Solver Jacobian method parameter is used to reduce this extra expensive cost for implicit solvers [24]. Also, an implicit solver requires fewer time steps [24]. And it requires fewer input derivatives, which is prompted by the simulation diagnostics of Simulink when failing to execute the simulator by employing an explicit solver. More information about explicit and implicit solvers can be referenced in Appendix A.6. The advantages of implicit solvers have been verified in the process of running the simulator.

As the result, the implicit solver, ODE14X is chosen. ODE14X is an extrapolation solver based on the linearly implicit Euler method [7]. It is the best global implicit fixed-step choice for physical systems, while it can fulfill the requirement of solution accuracy [22]. In this solver, the Number Newton’s iterations parameter, as shown in Figure 4-8, is to limit the number of global implicit iterations per time step.

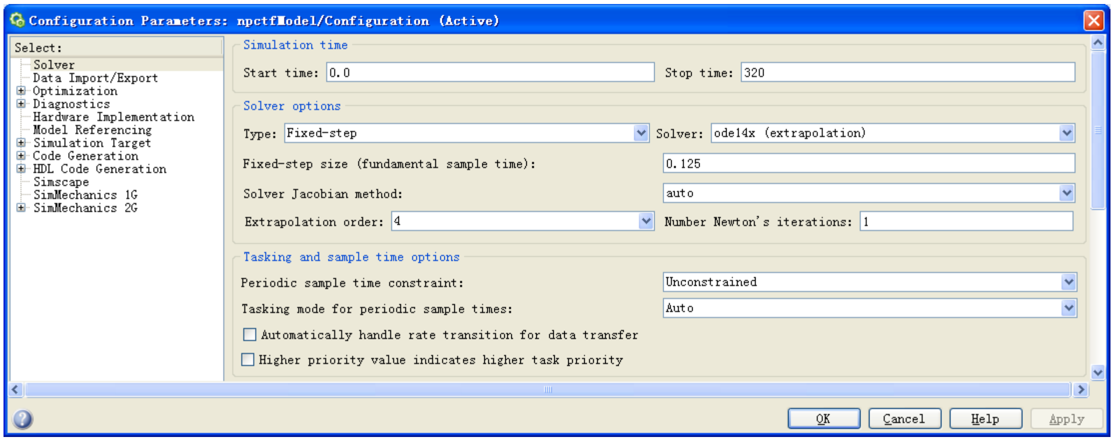


Figure 4-8 Model configuration of the simulator

4.5 Summary

The diagram of the simulator is constructed under the configuration and the Simscape scheme. As well as the built-in blocks, the custom-defined blocks are used in the diagram construction. The loop constructions as well as the block creations using the Simscape language are explained. In the simulator diagram, the monitoring methods, the simulation time, step size, and solver choice are introduced. The implicit fixed-step solver ODE14X is finally selected as the solver for the simulation system.

Chapter 5

5 Verification and Validation for the Simulator

Verification and validation (V&V) is the process of checking that the software system meets the specifications and that it fulfills its intended purpose.

5.1 Verification for the simulator

The simulation object is NPCTF. The scope described in the specifications has been satisfied. All the main components, valves, sensors, and loops listed in Table 1-1 are simulated in the simulator. The custom-defined components can be executed stably based on their complete Simscape codes. The constructed simulation loops have covered the desired simulation loops. The loops of filling water and draining water are added in the simulator diagram to complete the simulation functions.

The entire simulator can be executed without running issues. Simulation results can be exported by sensor blocks or monitoring blocks, and their physical units can be configured correctly. The parameters of the component blocks can be set in their dialogue boxes. During the simulation processes, the behaviours of the components in the loops can be simulated to demonstrate the thermal-hydraulic dynamic processes. The typical simulation functions are achieved, including:

- adjusting the flow rate of water and air by controlling valves;
- filling and draining water for the HX-Tank;
- controlling the input current to the Heater to adjust the input power;
- transporting temperatures among the Heater, Chiller, and Water Cooler to simulate heat transportations;
- controlling the flow rate of the pressurized air into the Pressurizer and the HX-Tank to alter their inner pressures and affect the water level in the HX-Tank;
- driving the Turbine to speed up with the pressurized air; and
- collecting simulation data from sensors and component outputs with time sequences for the next validations.

In summary, the simulator system meets its specifications. The simulation accuracy will be assessed in the following validations.

5.2 Validation step design

The validation for a simulation model provides assurance or confidence that the model is sufficient and appropriate for use according to the purpose of the model [3]. By evaluating the difference in the behaviours between the simulation models and the real systems, inconsistencies and errors can be rectified to make the models describe the real processes more accurately and realistically. The comparison between the results of the experiments and the simulations is herein used as a validation method.

5.2.1 Validation process

The following steps are conducted during the validation process: 1) collecting and processing experimental data; 2) applying the selected experiment data in the simulation models; 3) performing simulations on the simulator; 4) comparing the results of experiments and simulations; and 5) modifying models and parameters of the simulator to improve accuracy. Figure 5-1 illustrates the validation process.

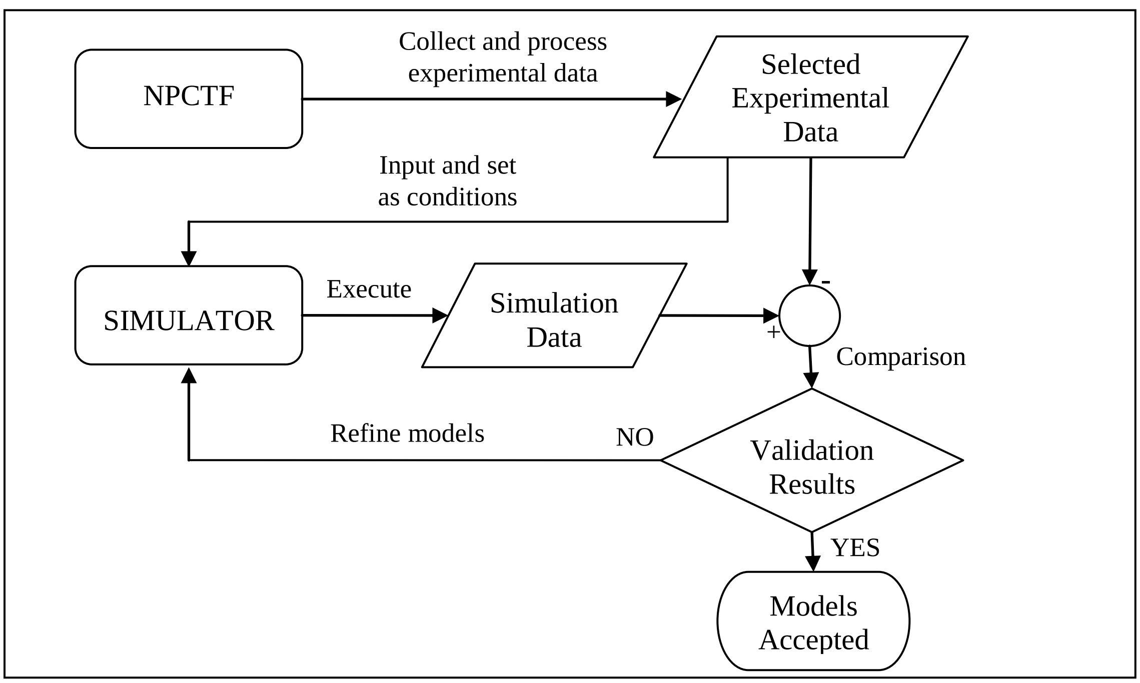


Figure 5-1 Validation process

The experimental data, or the measurement data, refers to the data obtained from the experiments on the physical facility. The experimental data are collected and processed before being used for model validations. In this process, data screening is used as the method by which to select the typical data including obvious dynamic characteristics. The experimental data are partitioned into two groups: 1) the conditions or inputs applied into the simulation models; and 2) the outputs applied to the comparison with the simulation results. The classification of the two groups depends on the purpose of the validation.

Signals from experiments are discrete. The sampling time interval in NPCTF is set to one second. The simulation system has a smaller fixed-step time, which is herein set as 0.125 s. To make sure the comparison can be conducted on the same time sequence, the output simulation data are trimmed to the same size as the experimental data.

5.2.2 Assessment figures

The absolute error and relative error of the simulation results are considered as the figures for assessing the quality of the simulation models.

1) The absolute error is defined as:

$$\varepsilon = N' - N \qquad (5\text{-}1)$$

where ε denotes the absolute error, N' is the simulation result, and N is the experimental result. The three parameters have the same physical units. ε may be positive or negative.

2) The relative error is defined as:

$$\delta = \frac{\varepsilon}{N} \times 100\% \qquad (5\text{-}2)$$

where the relative error δ is a dimensionless value expressed as a percentage (%).

5.2.3 Design of validation cases

The validations are conducted at the component level and the system level with quantitative assessment and qualitative assessment. Since sufficient validations of the components and systems are needed to achieve the validation goal, the schedules of validation cases are formed first. All the scheduled cases are designed in consideration of the common phenomena in the thermal-hydraulic processes of NPPs, which are listed in Appendix E.

At the component level, the focus is on the relationship of simple input variables and output variables. This is the base upon which to perform more complex simulations on more components and systems. Quantitative assessment is applied in the validation at the component level.

At the system level, tests for the behaviours of two or more components in one or more loops are conducted. These tests focus on the exhibitions of the components and the loops working together, in which multiple variables are involved. Since it is difficult to classify simple input and output variables, the qualitative assessment based upon model characteristics is more used.

The number of the scheduled validation cases is counted in Table 5-1. All the cases are explained as well as their validation results in the following sections.

Table 5-1 Number of validation cases

	Name	**Number**
Components	Heater	8
	Chiller	4
	Pressurizer	3
	HX-Tank	2
	Turbine	2
System	Primary coolant loop, Secondary water loop, Pressurized air loop	3
Total		22

5.3 Validation results of components

In this section, the validation results are presented in figure formats. All the simulations are conducted with same dependent parameters, including all the correlation coefficients of the Nusselt Number, the hydraulic resistances, and the pneumatic resistances.

In the following figures, the abbreviation *exp* represents *experiment* while *simu* represents *simulation*, and the figures of relative errors are with the unit percentages (%). The pressures indicated in the figures and tables are gage pressures if there are no special indications.

5.3.1 Dynamic processes in the Heater

There are eight typical cases selected to validate the dynamic responses of the Heater. All of these cases are related to the energy conversion in the Heater and are reflected on the outlet temperature of the working fluid. According to the mathematical model of the Heater, the outlet temperature, as a dependent variable, is influenced by the input variables, including the inlet temperature, flow rate of fluid, and input current. Since the Heater block is incorporated in the primary coolant loop, the inlet temperature and the flow rate are transported within the loop. To conduct the validation only on the Heater component, Test Mode is used by setting a mode input port in the Heater block and in some switch blocks. Under this mode, the experimental data can be copied into blocks of Lookup Table (n-D) and can then be input into the Heater block. The method is illustrated in Figure 5-2, in which the block of Clock can output the current simulation time at each simulation step [24], after which the blocks of Lookup Table can output data according to the array of experiment data and simulation time.

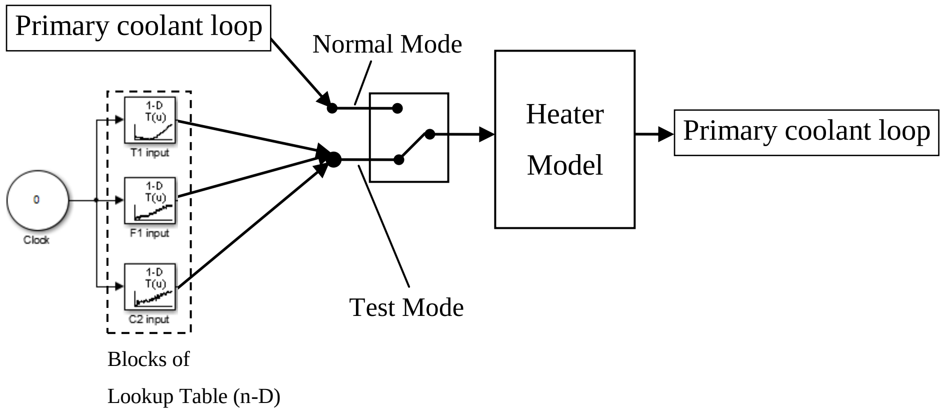


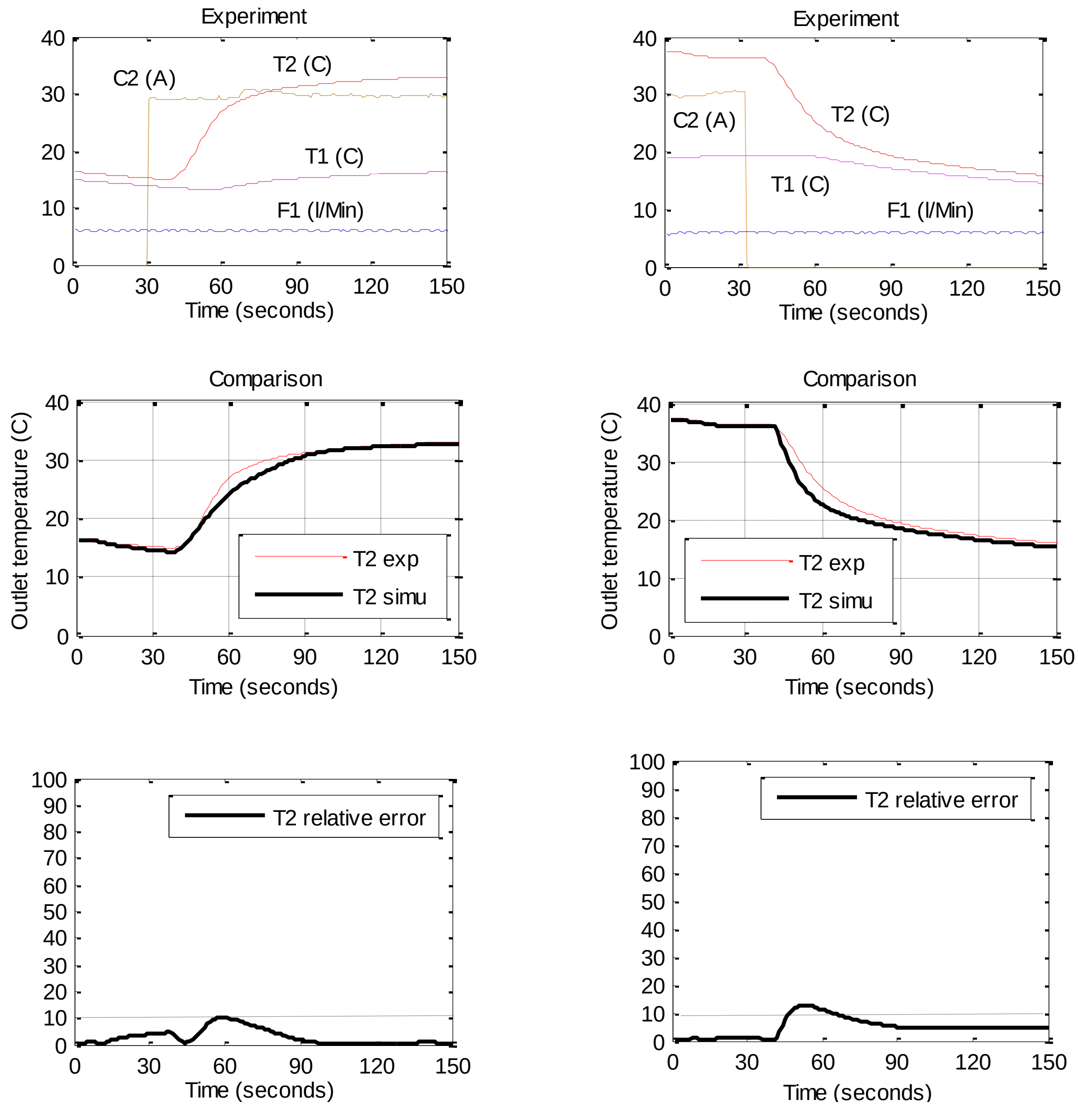
Figure 5-2 Illustration of the validation method of the Heater

Based on its mathematic model, the other parameters with constants are also involved in the calculations. In the process of heat transfer, the correlation coefficients of the Nusselt number influence the convection heat transfer coefficients between the fluid and the metal. These correlation coefficients are tested according to the experiments. And they are maintained as same values for all the validations.

In the eight cases, the disturbances are from the current, the flow rate, and the inlet temperature. The rising and dropping of the input variables are treated as distinct disturbances in the different cases. The first six cases are mainly performed with one distinct disturbance. Then, in the following three cases, two obvious disturbances from two variables are tested. All the disturbances affecting the outlet temperature are summarized as: 1) current rising; 2) current dropping; 3) flow rate rising; 4) flow rate dropping; 5) inlet temperature rising; 6) inlet temperature dropping; 7) increasing current and flow rate; and 8) decreasing current and flow rate.

1. Response of current rising and dropping to outlet temperature (Case 1.1, Case 1.2)

The process of energy conversion is expressed as the response from the current C2 to the outlet temperature T2. While the water flow rate F1 and the inlet temperature T1 remain smooth, the power is generated and then is transferred to the fluid through the metal sheath covering the electrical heating rod. In the dynamic model, as presented in Chapter 2, three differential equations represent the dynamic process.



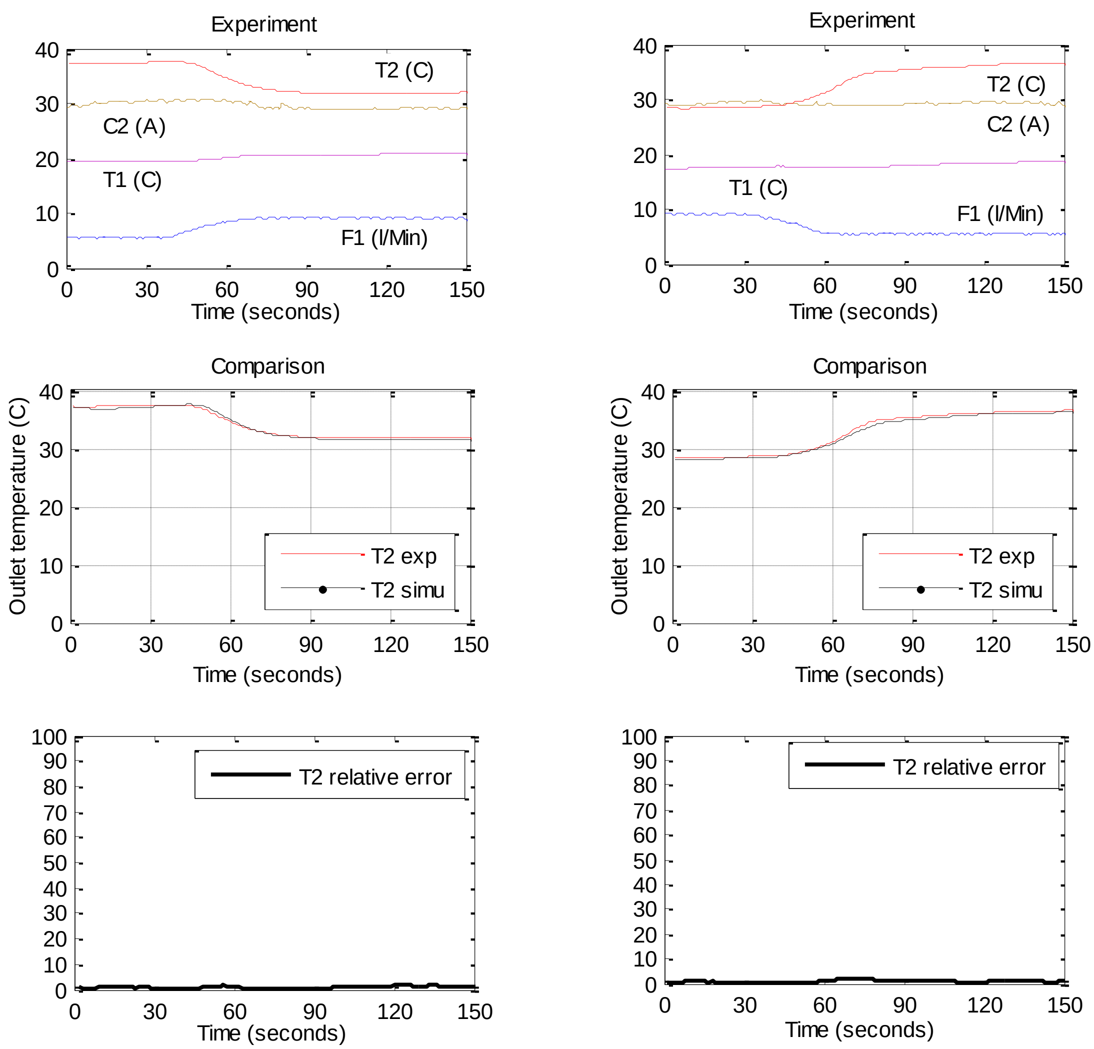
Case 1.1: Current rising Case 1.2: Current dropping

C2 (A): current F1 (l/Min): water flow rate T1 (℃): inlet temperature T2 (℃): outlet temperature

Figure 5-3 Response of the current change to the outlet temperature in the Heater

As the figures show, the obvious errors between the measured temperature and the simulation temperature occur in periods of C2 rising and dropping sharply. Most of the relative errors are maintained below 10%. However, the maximum of the errors above 10% occur in the C2 dropping process in case 1.2. The reason causing the errors may come from the distribution characteristics of the temperatures of the water and the metal wall. In the lumped parameter models, the entire inner water as well as the metal sheath is assumed to have uniform temperatures and properties. This assumption may make the outlet temperature in the simulation response more quickly than the real situation under the input current varying dramatically. This quick response then can result in the large errors, especially in the current dropping process.

2. Response of flow rate rising and dropping to outlet temperature (Case 1.3, Case 1.4)



Case 1.3: Flow rate rising Case 1.4: Flow rate dropping

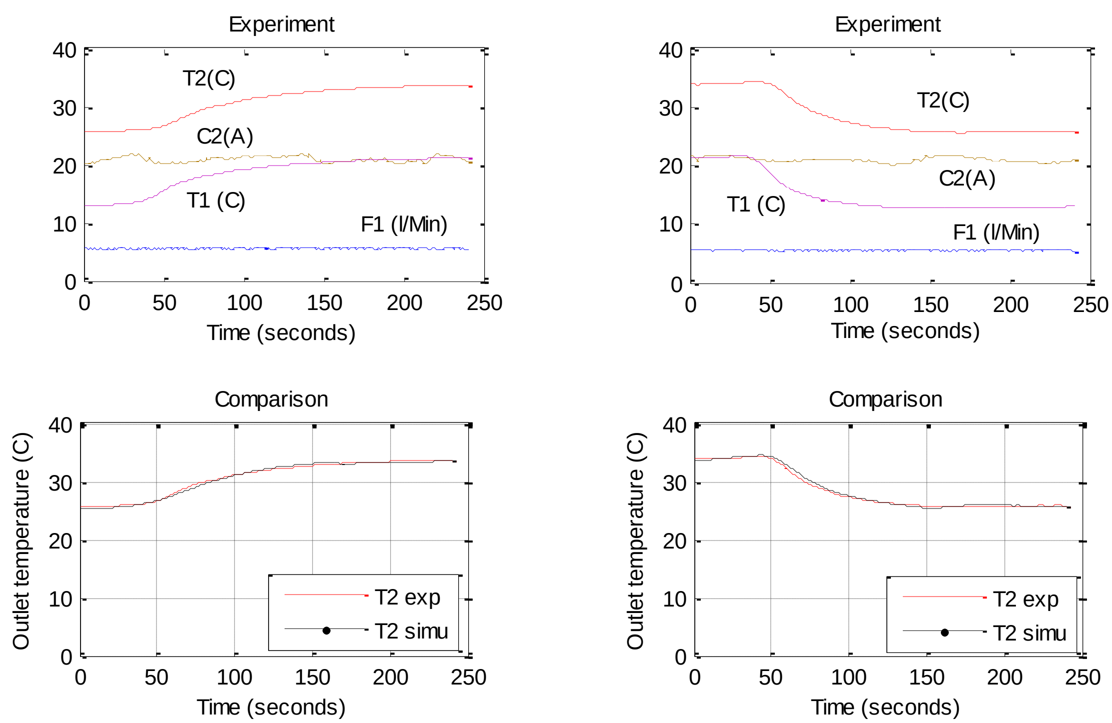
C2 (*A*): current F1 (*l/Min*): water flow rate T1 (℃): inlet temperature T2 (℃): outlet temperature

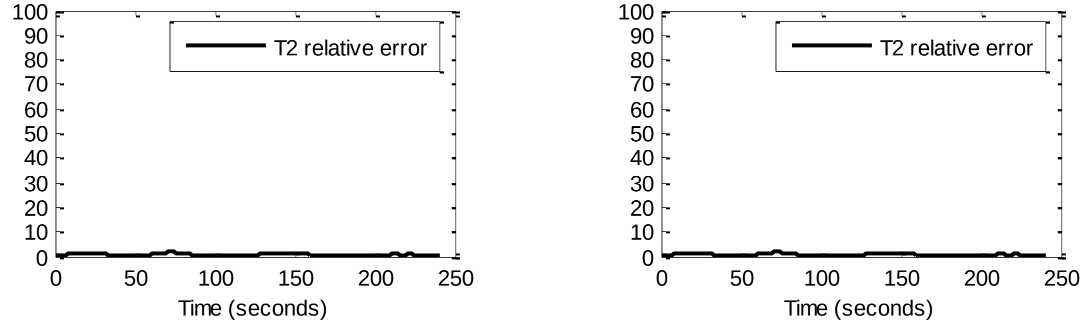
Figure 5-4 Response of the flow rate change to the outlet temperature in the Heater

The two cases perform the validation on the effect from the flow rate F1 to the outlet temperature T2. The results show that there is quite a good agreement between the simulations and the experiments. The relative errors of T2 are almost under 2%. It means that under the conditions of relatively smooth C2 and T1, the response from F1 to T2 is very close between the simulations and the experiments. The reason is that this response is mainly based on the dynamic process of the water side, which is expressed as the one differential equation in the Heater model.

3. Response of inlet temperature increasing and decreasing to outlet temperature (Case 1.5, Case 1.6)

The two cases test the response from the inlet temperature T1 to the outlet temperature T2. Similar to the effect of F1 to T2, the response from T1 to T2 in the simulations matches well with the experiment results. The relative errors are mostly below 3%.



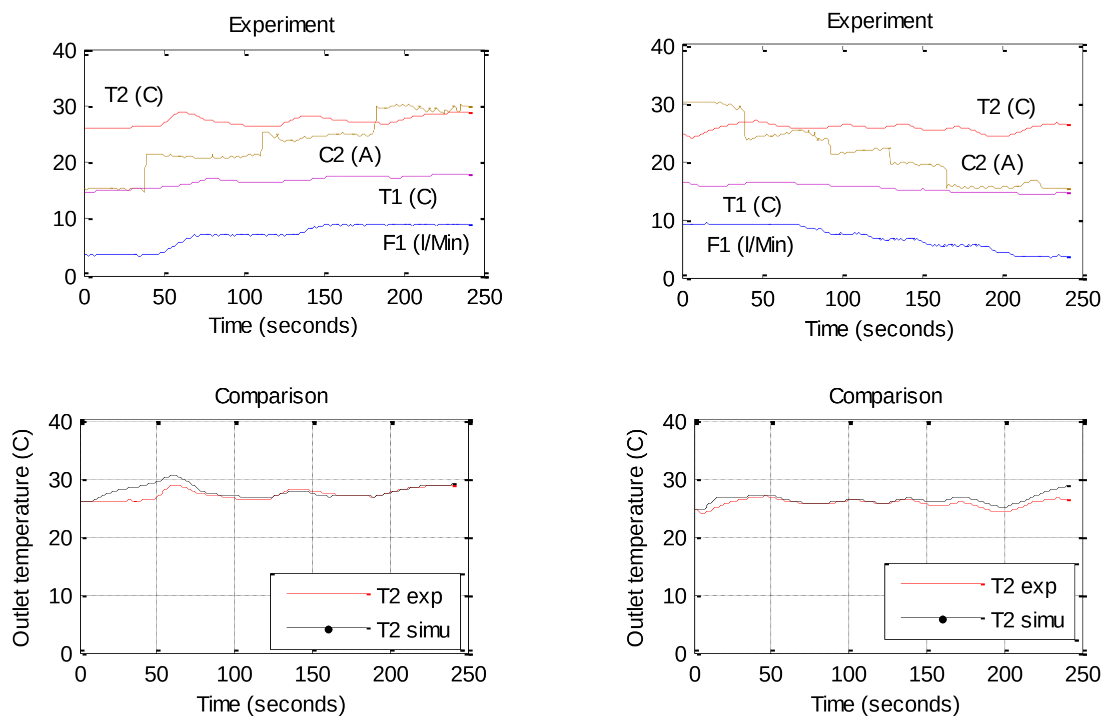


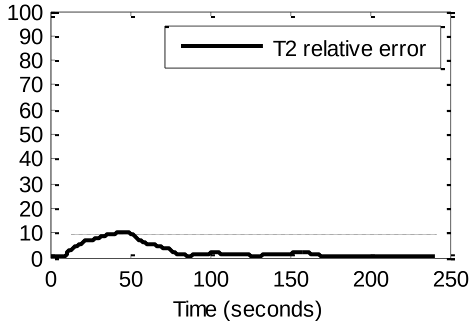
Case 1.5: Inlet temperature increasing   Case 1.6: Inlet temperature decreasing

C2 (*A*): current   F1 (*l/Min*): water flow rate   T1 (℃): inlet temperature   T2 (℃): outlet temperature

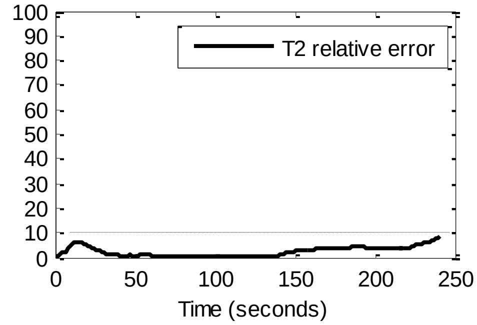
Figure 5-5 Response of the inlet temperature to the outlet temperature in the Heater

4. Response of changes of current and flow rate in parallel to outlet temperature (Case 1.7, Case 1.8)





Case 1.7: Current and flow rate increasing



Case 1.8: Current and flow rate decreasing

C2 (*A*): current F1 (*l/Min*): water flow rate T1 (℃): inlet temperature T2 (℃): outlet temperature

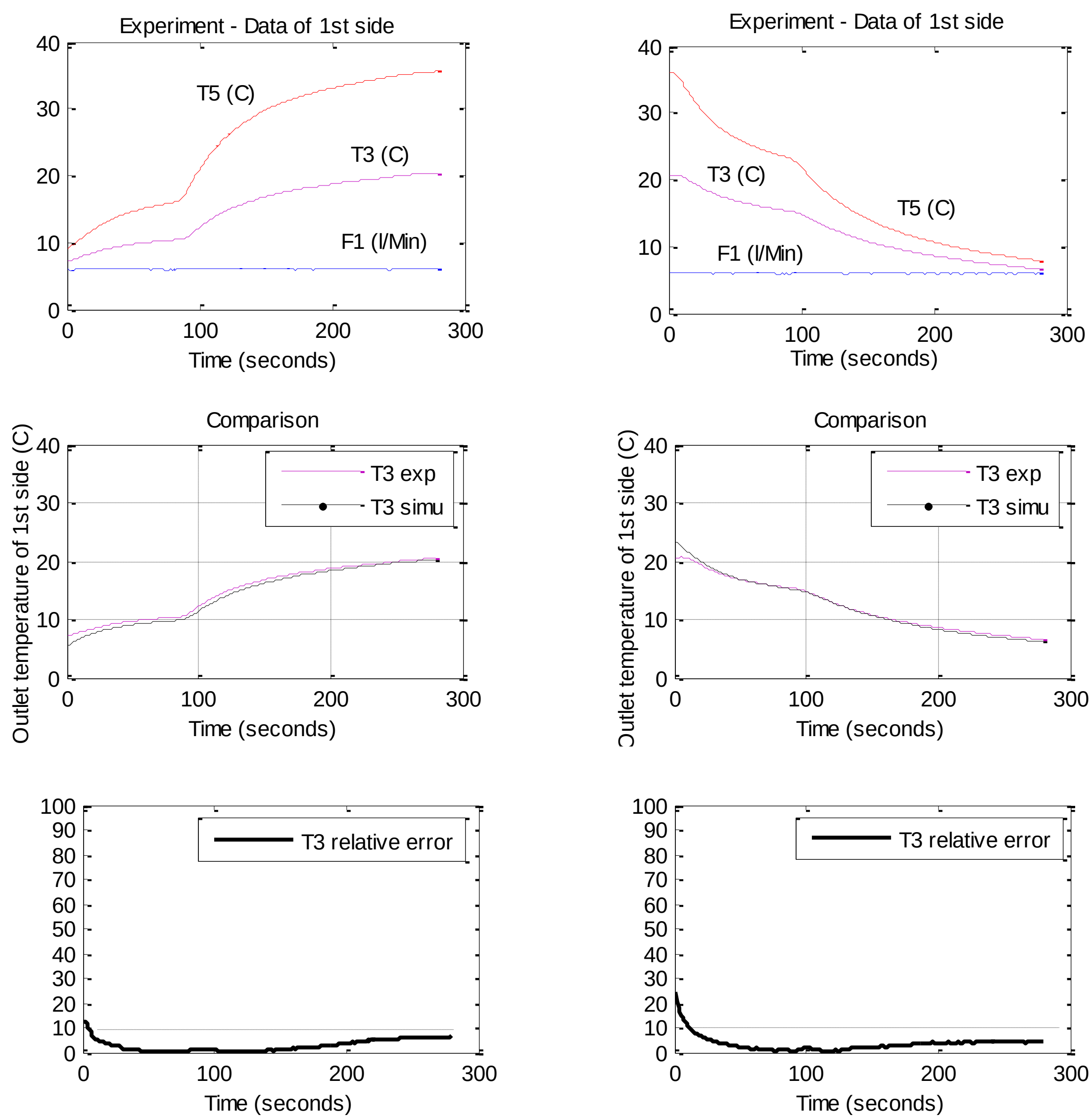
Figure 5-6 Response of the current and flow rate to the outlet temperature in the Heater

Two disturbances of the current C2 and the flow rate F1 act together on the outlet temperature T2. Actually, for a device of energy conversion such as a reactor or a heater, the input power and the working fluid can work together to maintain the outlet thermal status at a stable line. In the operations, F1 is adjusted repeatedly to compensate for changes of C2. The results can be considered as a sum of the effects of the current and flow rates. The T2 errors are obviously large in the first 40 seconds where there is no flow rates to be adjusted. This first process is similar to the dynamic response of the current. Next, the flow rate F1 is involved in the operations, which makes the T2 errors decrease to a lower level. Therefore, the current action should account for the main errors between the simulations and experiments.

5.3.2 Dynamic processes in the Chiller

In Chiller, there are two sides, referred to as the primary side (1st side) and the secondary side (2nd side). Since the chiller pump (Pump4) and the Water Cooler are powerful enough to transport the heat to the ambience on the second side, and the temperatures in the Water Cooler are not obtained, the validations of the Chiller are mainly conducted on the primary side. The validation method is similar to that of the Heater. Four typical validation cases are used to test the responses of the inlet temperature and flow rate to the outlet temperature. These cases are explained in two groups as follows.

1. Response of inlet temperature varying to outlet temperature (Case 2.1, Case 2.2)



Case 2.1: Inlet temperature increasing   Case 2.2: Inlet temperature decreasing

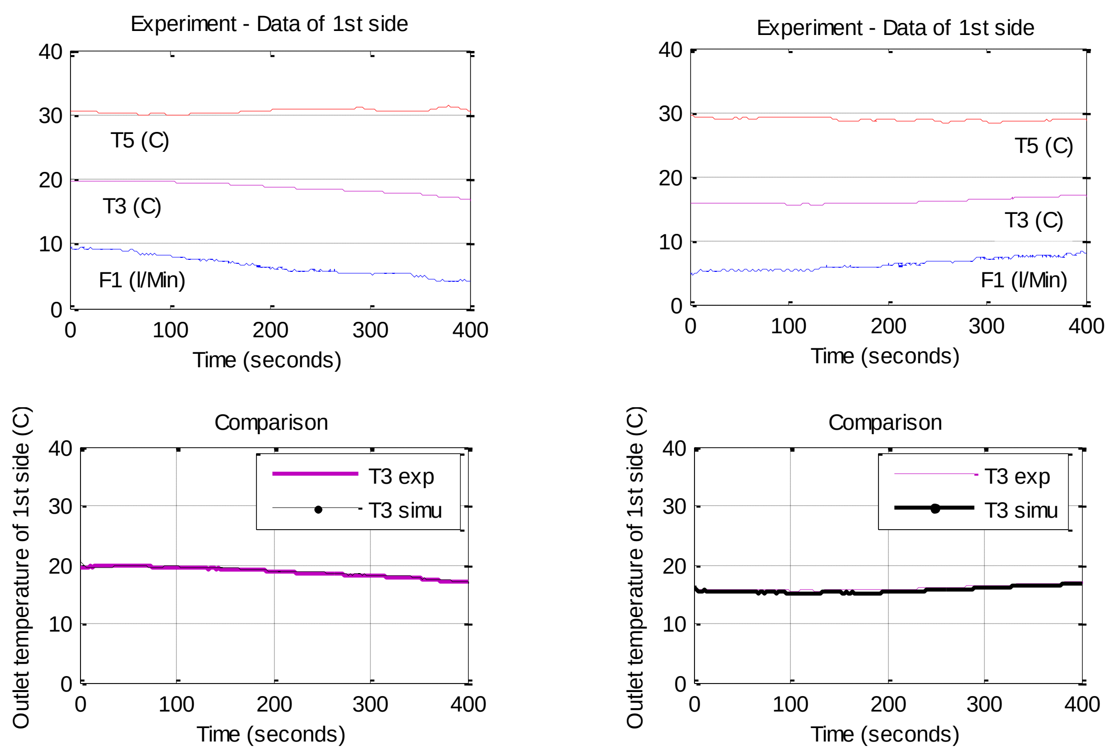
F1 (*l/Min*): water flow rate	T5 (℃): inlet temperature	T3 (℃): outlet temperature

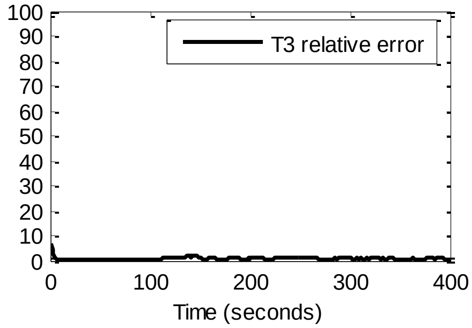
Figure 5-7 Response of the inlet temperature varying to outlet temperature in the Chiller

The two cases are used to check the response from the inlet temperature T5 to the outlet temperature T3 on the primary side of the Chiller. T5 varies in scope from about 8 ℃ to 36 ℃. T3 then varies from about 7 ℃ to 21 ℃. The T3 errors in first 20 seconds are significantly larger. This is due to the heat balancing process of the Water Cooler. In simulation, the Water Cooler is set with initial conditions with the ambient temperature, therefore the secondary side of the Chiller is started with a higher temperature, and the temperature of the primary side also starts with a relatively high temperature. This is different from the experiment. As a result, in the first 20 seconds, the errors of T3 show larger than the following. This is a drawback of this simulator.

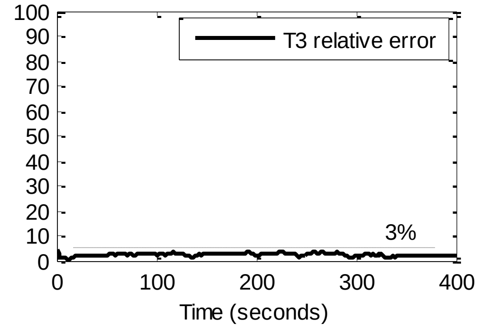
The temperatures use Celsius scale. In the above cases, the temperatures can reach relatively lower degrees, such as 7 ℃. This makes the relative errors might achieve higher values. If using thermodynamic temperatures, the relative errors can be reduced to lower levels with the same absolute errors. However, since the absolute errors can also be reduced to smaller values in the simulations, the calculations for the temperature errors in Celsius scale can be still valid, and the relative errors are then used herein as the figures for the assessment.

2. Response of flow rate varying to outlet temperature (Case 2.3, Case 2.4)





Case 2.3: Flow rate decreasing



Case 2.4: Flow rate increasing

F1 (*l/Min*): water flow rate T5 (℃): inlet temperature T3 (℃): outlet temperature

Figure 5-8 Response of the flow rate varying to outlet temperature in the Chiller

These two cases validate the response from the flow rate F1 to the outlet temperature T3 on the primary side in the Chiller under the relative stability of the inlet temperature T5. The disturbances are mainly from F1 rising and falling. Since the secondary side of the Chiller is linked to the strong cooling circuit with the Water Cooler, the varying of the flow rate has a small effect on the outlet temperature.

Very similar to the Heater model, the effect of the flow to the heat exchange is in better agreement between the simulations and the experiments. Most of the relative errors of the output temperature are maintained below 3%.

5.3.3 Dynamic processes in the Pressurizer

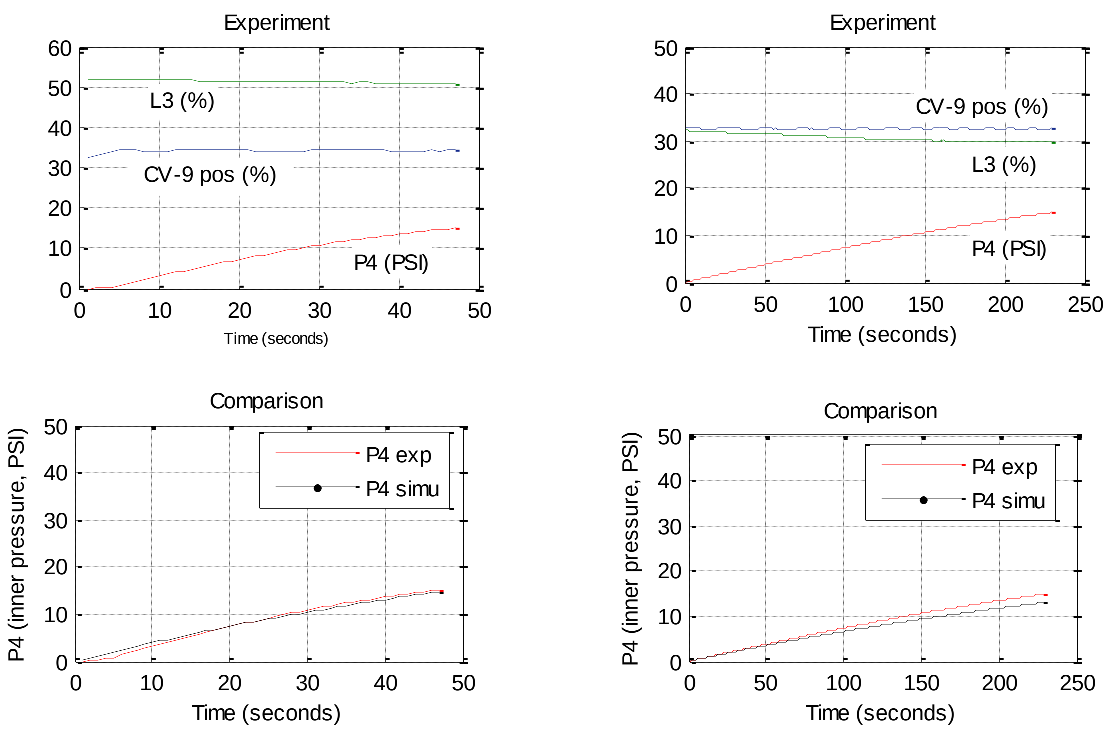
The Pressurizer works with two fluids: water and air. In NPCTF, the measured variables are only the water level L3 and the inner air pressure P4. There are no flow meters for water and air. Hence, the positions of air valves are employed as the validation conditions for the air flow rates. It is assumed that the relationship of the flow rate and the valve position is stable on the valve curves.

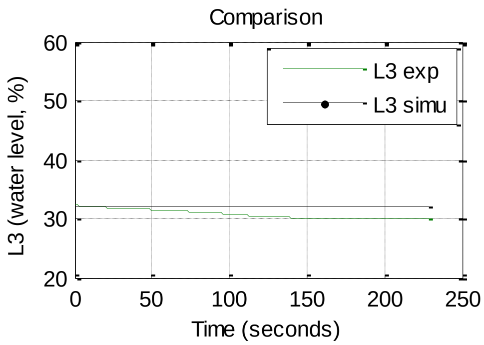
Three validation cases are conducted by testing the effects from the air inlet valve CV-9 and the air release valve CV-10 on the inner air pressure P4 and the water level L3. In simulations, the positions of the air valves are input by the block of Lookup Table blocks with the experimental data.

1. Opening air inlet valve CV-9 to raise inner pressure (Case 3.1, Case 3.2)

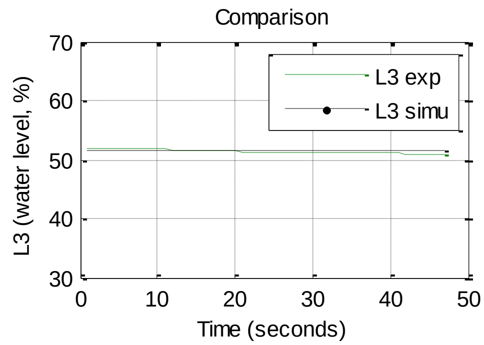
Two cases are conducted to check the response of opening the air inlet valve CV-9 to the inner air pressure P4. The conditions of the water level L3 of case 3.1 and case 3.2 are 52% and 32%, respectively. The sensitive position of CV-9 is about 32%–33%. Near to this position, the speed of P4 rising varies significantly owing to the flow rate varying.

The experiment results show that P4 starts to rise with CV-9 opening at the position over 32%. At this position, CV-9 just overcomes its dead zone, and the air flow can pass through the valve. In experiment, the change of the sound of air flow can be heard with it increasing.





Case 3.1: Open CV-9 under L3=52%



Case 3.2: Open CV-9 under L3=32%

P4 (*PSI*): inner air pressure L3 (*%*): water level

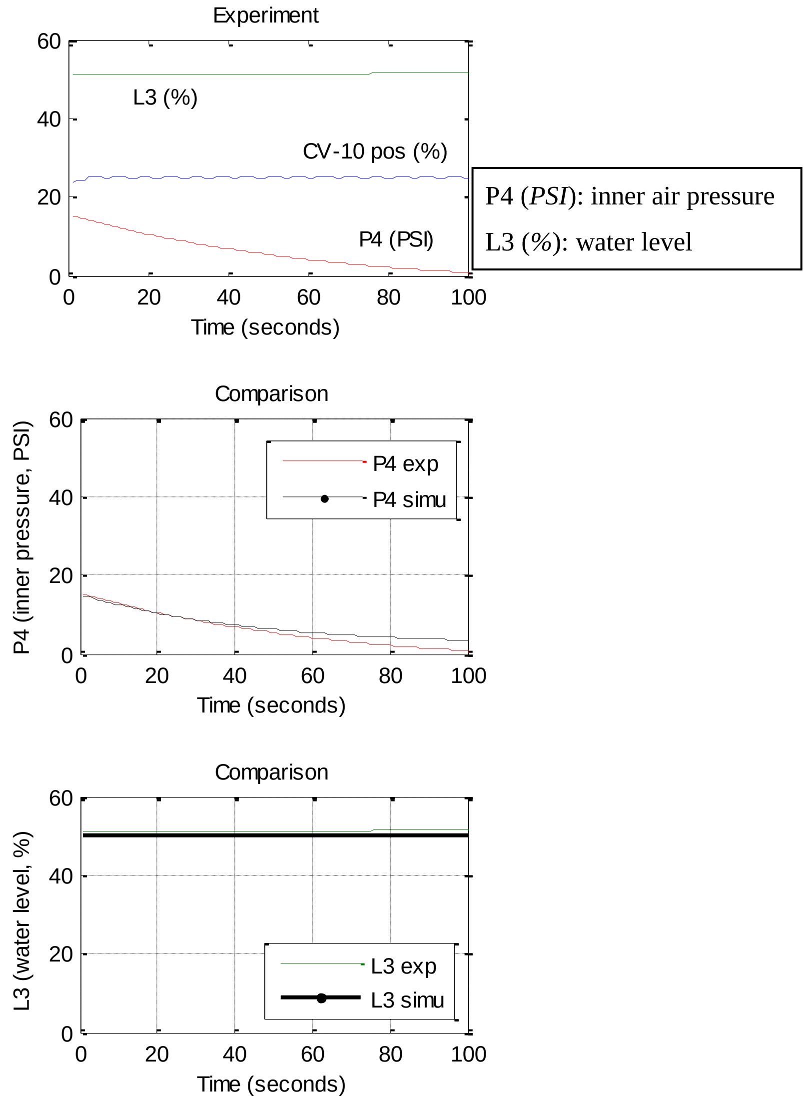
Figure 5-9 Opening the inlet valve in the Pressurizer

The two cases are conducted in two different experiments under different conditions. Although P4 rises in Case 3.1 more quickly than in Case 3.2, it is not concluded that this is owing to the water level L3. The validations are just to test the responses of the inner pressure to changes in the valve position as well as the air flow rate. The errors of the simulations are small. Measurement errors may be from the feedbacks of valve positions and the pressure sensor, which may contribute to the simulation errors.

The water levels of the simulations show more smoothly than those in the experiments. This may reflect ideal situations. In the real system, the water is not exactly incompressible, and the loops are not 100% sealed. Furthermore, since the measured water level is from the difference of the upper and lower pressures, the inner pressure may influence the measurement of the water level.

2. Opening air release valve CV-10 to release inner pressure (Case 3.3)

This case is conducted to test the response of the inner air pressure P4 when releasing air. When opening the air release valve CV-10, the air in the Pressurizer is released to the atmosphere.



Case 3.3: Open CV-10 under L3=50%

Figure 5-10 Opening the release valve in the Pressurizer

In the simulation, the air inlet flow rate cannot be reduced to zero when fully closing the inlet valve CV-9. This can contribute to the simulation errors. Especially at the end of the simulation, this causes P4 to be higher by about 2 *PSI*.

In mathematical models, the compressed air is considered as an ideal gas. Therefore, there is a linear relationship of air density and air pressure based on unchanged temperatures. This means the values of air pressure are only associated with the values of air density, which is very similar to saturated steams.

5.3.4 Dynamic processes in the HX-Tank

Case 4.1 is designed to check the relationship of the inlet valve position and the air pressure as well as the influence of air bubbles on the water level. Case 4.2 is used to check the water filling process. The processes of adjusting inner air pressure and filling the water are the main duty of the HX-Tank.

1. Opening air inlet valve CV-21 to raise inner pressure (Case 4.1)

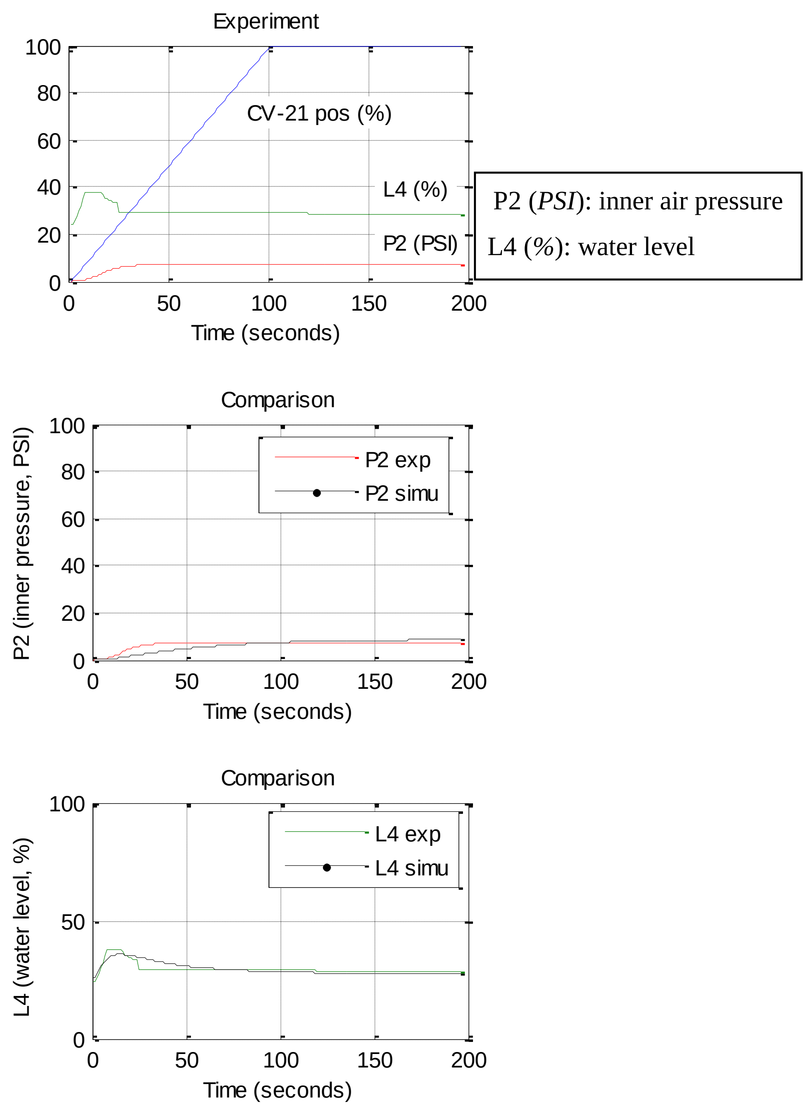


Figure 5-11 Opening the inlet valve in the HX-Tank

In the simulation, P2 rises a little more slowly than in the experiment. This means that the calculated air flow in the simulation is less than that enters the tank at first in the experiment. Also, L4, which is influenced by the bubbles, shows much more smoothly in the simulations. This may be due to the continuous iteration of the bubble volume.

2. Opening water inlet valve CV-25 to fill water (Case 4.2)

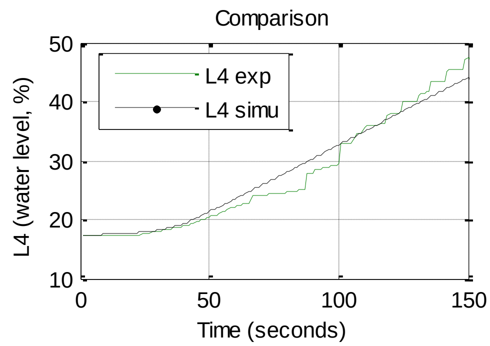


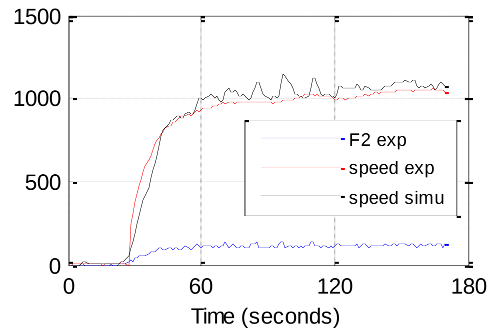
Figure 5-12 Filling water into the HX-Tank

This experiment is conducted by opening the water inlet valve CV-25 from 20% to 80%, while maintaining the water outlet valve CV-18 at 30%. There is no measurement for the water flow rate; thus, the comparison is just conducted with respect to the water level L4. In the simulation, the inner air pressure is not at zero, and the air pressure can affect the amount of water filling in and draining out. Despite this, the water levels of the simulation and experiment match well.

5.3.5 Dynamic processes in the Turbine

Two validation cases are designed to identify the parameters of the speed equation and to check the relationship of the inlet valve position and air flow rate as well as air pressure.

1. Relationship of the air flow rate and the speed (Case 5.1)



F2 (*l/Min*): air flow rate	speed (*rpm*): draft speed

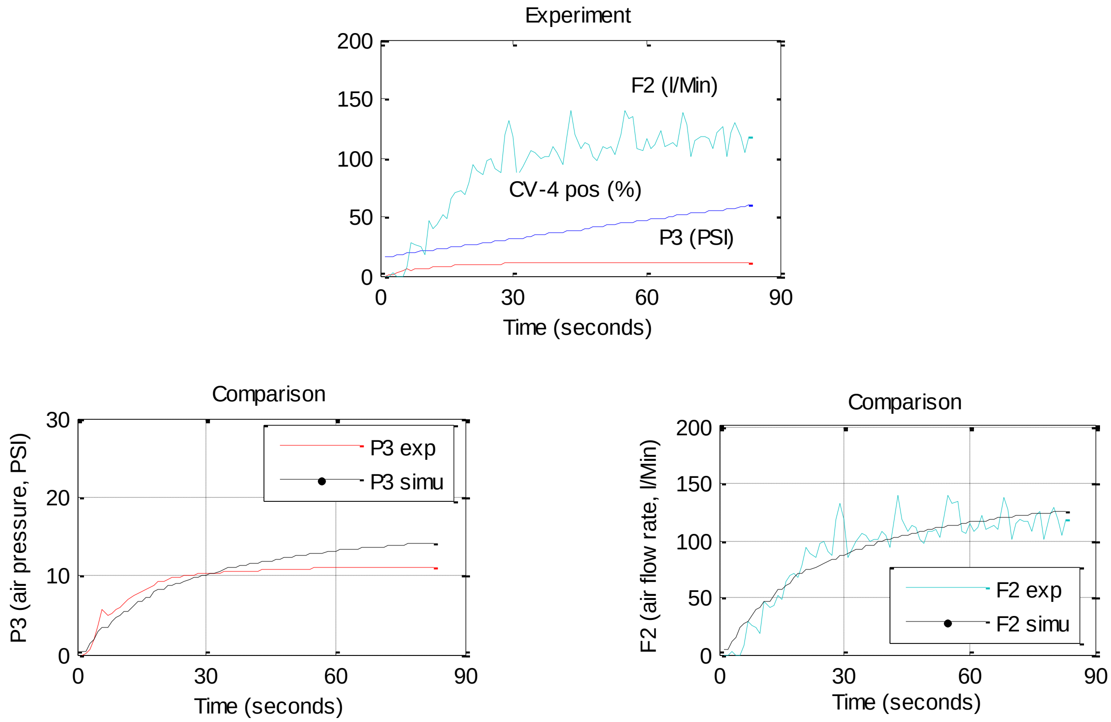
Figure 5-13 Relationship of the air flow rate and the speed in the Turbine

The inlet air flow rate F2 and the shaft speed are measured in NPCTF. In the simulation, the measured data of F2 is input with the block by Lookup Table under Test Mode, which is like the method in the Heater. By identifying the suitable parameters, including the moment of inertia and the friction factor, the trend of the simulated speed matches well with the experimental observation.

The erratic fluctuations of the shaft speed in the simulation are caused by the input flow rate with experimental data. The air flow from the experiment shows fluctuated due to the measurement. Indeed, it is smoother in driving the turbine shaft running. Thus, when the measurement data of air flow are used as the input into the Turbine model, the simulation results of the output shaft speed show fluctuated, too. This is the source of the unavailability of the quantization errors.

2. Opening air inlet valve CV-4 to raise pressure and flow (Case 5.2)

Case 5.2 uses a method similar to that for the Pressurizer to assess the response of the air flow rate F2 and inner air pressure P3 when the inlet valve CV-4 opens.



P3 (*PSI*): air pressure after inlet valve	F2 (*l/Min*): air flow rate

Figure 5-14 Opening the air inlet valve in the Turbine

Case 5.2 checks the relationship from the inlet valve CV-4 position to the air flow rate F2 and the air pressure P3. The results exhibit the same trend consistency between simulation and experiment. However, the errors of P3 and F2 show a bit large, which may be caused by the factors including: a) the measurement errors of the air pressure and air flow rate in experiments; b) the discrepancy of initial conditions of the air loop between experiment and simulation; and c) the supply air pressure of the air system in the simulation model. The supply pressure of the compressed air is set as a constant in order to ensure the air loop can work normally. In fact, this supply air pressure may vary a bit owing to the influence from the consumed amount of the subsystems of the Pressurizer, HX-Tank, and Turbine. All these factors cause the simulation errors to be a little large.

5.4 Simulation and validation at the system level

The validations of the dynamic behaviours of the main components are intended to check the response from the input variables to output variables of the component models. The input variables can be treated as independent variables for the dynamic processes in the components. The output variables are determined only by the input variables and the inner parameters. However, in the executions of the entire simulator system, the independent variables are only those initial parameters providing the initial conditions for the loops and the components. Therefore, the validations at the system level are followed closely by simulation characteristics of dynamic behaviours of the entire system.

5.4.1 Heat transportation process

Case 6.1 is to check the heat transportation process between the Heater and the Chiller. As Figure 5-15 shows, four temperature sensors are placed in this loop. The thermal capacities of the pipes between the two components cause temperature delays. In the simulator, this process is demonstrated through two blocks of Thermal Inertia.

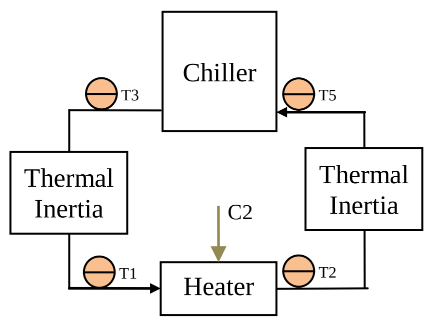
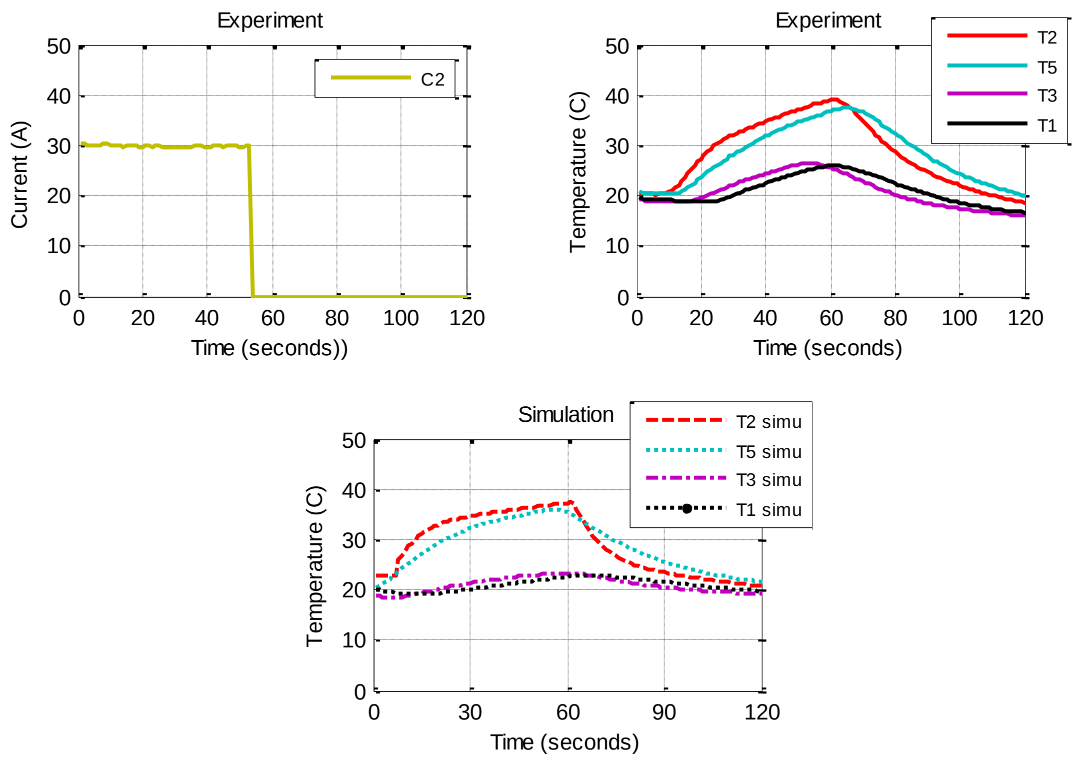


Figure 5-15 Thermal process between the Heater and the Chiller



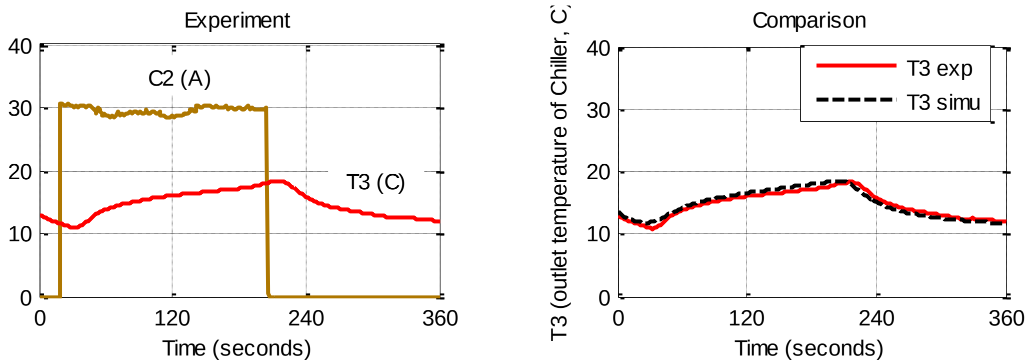
C2 (*A*): current input into the Heater	
T1 (℃): inlet temperature of the Heater	T2 (℃): outlet temperature of the Heater
T5 (℃): inlet temperature of the Chiller	T3 (℃): outlet temperature of the Chiller

Figure 5-16 Temperatures between the Heater and the Chiller

Figure 5-16 shows the results from the experiment and the simulation. The rising temperatures are caused by the input current C2 of the Heater. In the simulation, C2 uses experiment data as the input to the Heater, in which the method is the same as the component validation in the Heater. By identifying the appropriate equivalent structure sizes in the thermal inertia blocks, the simulation temperatures exhibit behaviours similar to those in the experiments.

5.4.2 Energy balance

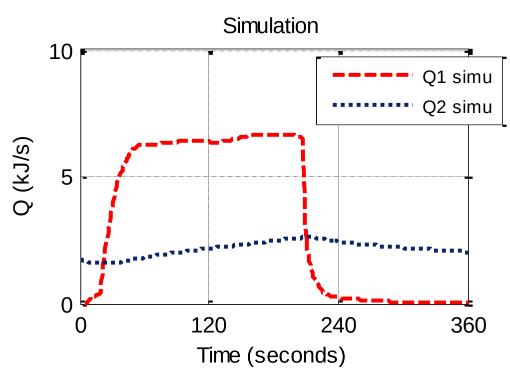
Case 6.2 is a test of the energy balance in the Heater and the Chiller. As the heat is generated and released in the two components, like the processes in a reactor and steam generator in an NPP, the thermal balance can be exhibited by the outlet temperatures.



C2 (*A*): current input into the Heater   T3 (℃): outlet temperature of the Chiller

Figure 5-17 Energy balance in the Heater and the Chiller

In this case, T3 is employed as the assessment variable to validate the models. The conditions of the experiment and the simulation are not exactly the same. The results of T3, however, exhibit a good consistency with small errors, as shown in Figure 5-17.



Q1 (*kJ/s*): heat carried out of the Heater   Q2 (*kJ/s*): heat released on the secondary side of the Chiller

Figure 5-18 Generated heat and released heat in the simulation

As a typical example, Figure 5-18 shows the heat carried out of the Heater and the heat released on the secondary side of the Chiller in the simulation. These two variables, as well as other more internal variables, are calculated in the simulation models. These variables are not measured in NPCTF. However, they can be obtained from the simulator for analysis. This is an advantage of a software simulator.

5.4.3 Momentum balance

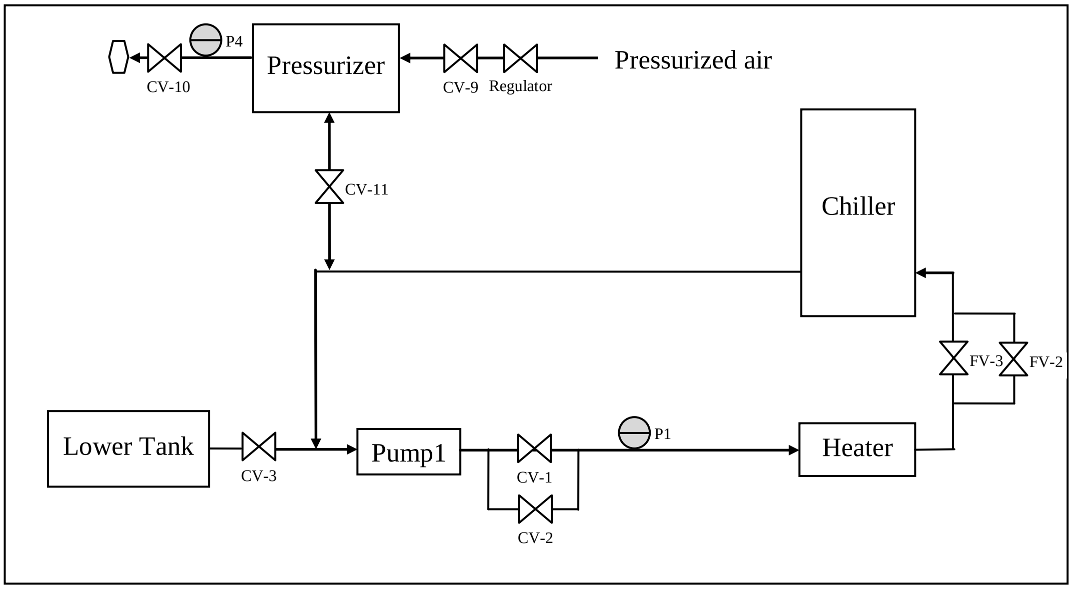
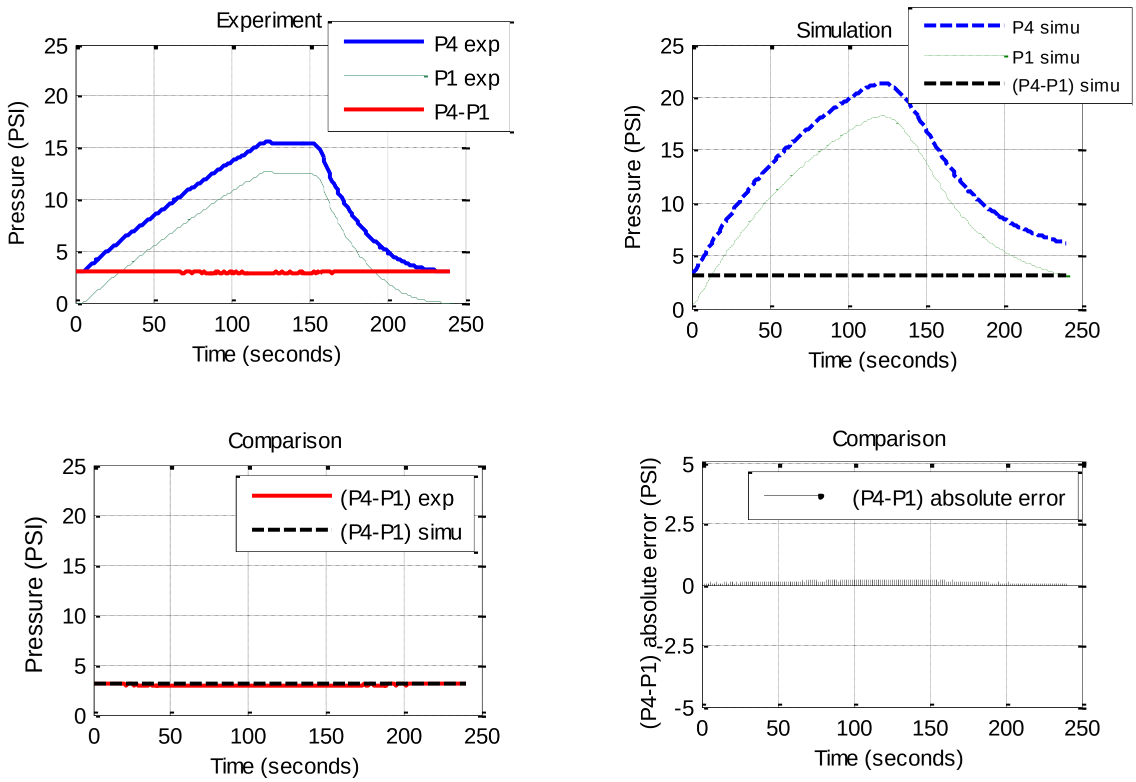


Figure 5-19 Pressurizer and the primary coolant loop

Case 6.3 is used to validate the operating mode, “WITH PRESSURIZER.” This system diagram is shown in Figure 5-19. When the connect valve CV-11 of the primary coolant loop and the Pressurizer is opened, the inner air pressure of the Pressurizer (P4) can affect the pressure of the primary loop (P1). In NPCTF, P4 and P1 work in a scope under 18 *PSI*, where 18 *PSI* is the trigger pressure that can initiate the shutdown system of NPCTF. Moreover, the process of pressures rising in the simulation cannot be controlled along the same curve of the experiment. This is a difficult issue in this validation case. However, since the difference of the two pressures, that is P4-P1, should be a constant, theoretically, the validation focuses on the difference of the two pressures.



P1 (*PSI*): water pressure of the primary coolant loop   P4 (*PSI*): air pressure of the Pressurizer

Figure 5-20 Pressure balance in the Pressurizer and the primary coolant loop

In the experiment, as the above figure shows, there is a flat during the pressures rising process. This is the result of closing the air inlet valve CV-9, which can make P4 avoid reaching its high limit. In the simulation, however, CV-9 has to be maintained open and P4 reaches higher pressures, which is due to the difficulties of inserting the operations at the same time as the experiments on this simulation platform. On the other hand, the pressure difference (P4-P1) remains almost constant for the entire duration of the process. Some low amplitude fluctuation around a constant value is shown in the experiment. The fluctuation could be a result of random errors in the measurements. The absolute errors of the pressure difference (P4-P1) are small, mostly within 0.2 *PSI*.

5.5 Simulation case study

As a further test under the above assessments, a severe simulation case is designed to study the scenario in the Heater and the Chiller when the input current of the Heater rises sharply and exceeds its limit in NPCTF.

As Figure 5-21 and Figure 5-22 show, the water flow rate in the Heater and that on the primary side of the Chiller, G and G1, remain at 6 *l/Min*. The water flow rate on the secondary side of the Chiller, G2, remains at nearly 58 *l/Min*. In NPCTF, the current, C2, is limited within 30 *A*. In this simulation, C2 stays at 30 *A* in the first 240 seconds, and then rises double to 60 *A*. This causes the temperatures in the loop to rise significantly, especially the outlet temperatures of the Heater and the Chiller, T2 and T3.

This scenario simulates the dynamic process initiated by the significant rise of the input power under severe accident conditions.

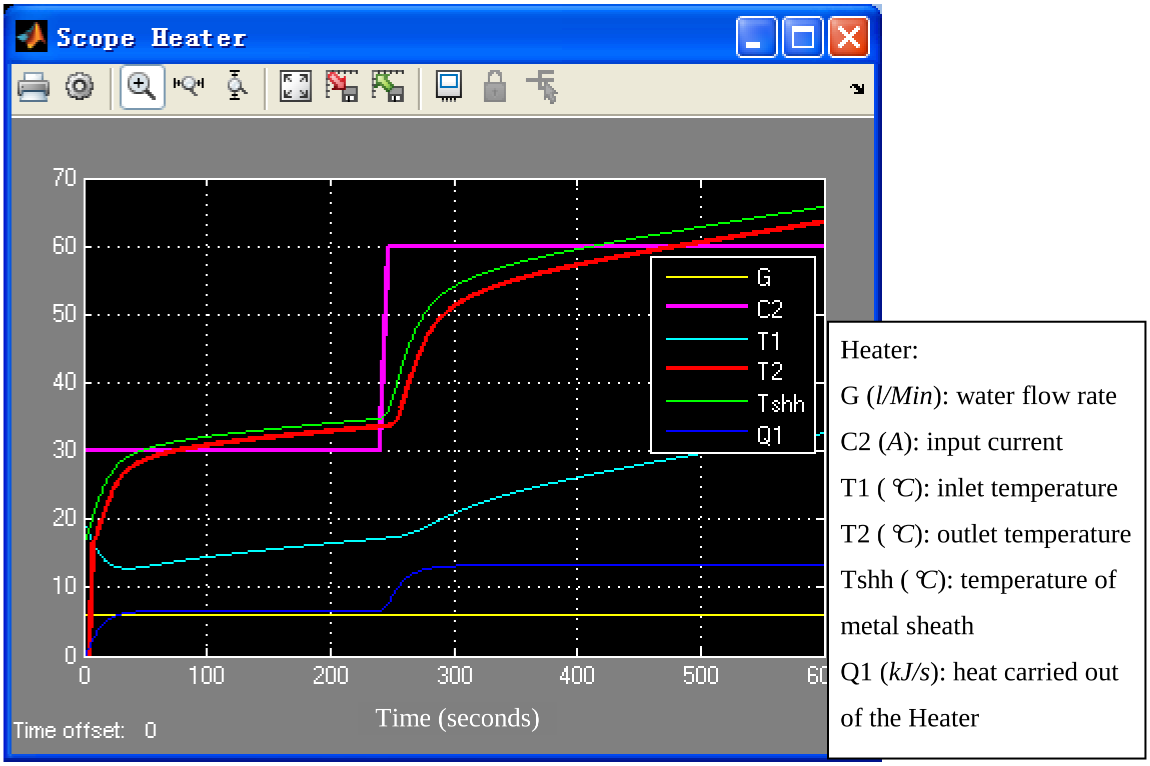


Figure 5-21 Monitoring of variables in the Heater

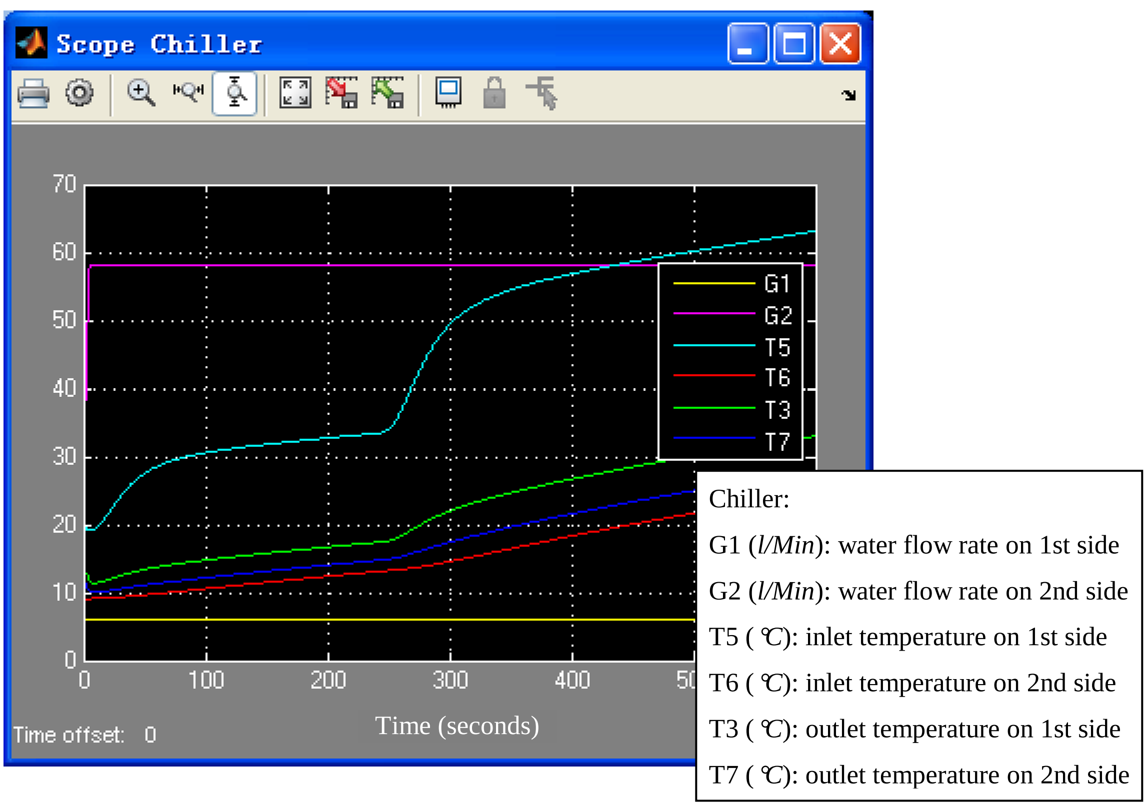


Figure 5-22 Monitoring of variables in the Chiller

5.6 Summary of validations

A series of validations have been conducted with the experimental data and simulation results. The exhibitions of the simulator are assessed by comparing the outputs of the experiments and simulations.

All the validation cases actually validate the balances of mass, momentum, and energy. The validation cases in the Heater and the Chiller check the energy balance in terms of heat exchange and heat transfer. In the Pressurizer and the HX-Tank, the mass balances of the fluids are examined by controlling the valves to change the pressures and water level. The validation cases of the Turbine are intended to investigate the momentum balance through the relationship of flow rate and shaft speed.

The simulation errors can be viewed from two viewpoints: 1) the simplification of the models and 2) the discrepancies in the measurements. Simplification of the models may be the main reason for most simulation errors. The uncertainties of the experiments, such as varying ambience or the shifts of sensors, also affect the accuracy of the experimental data as the benchmark of the validations.

Chapter 6

6 Conclusion, Summary, and Future Work

This thesis presents the modeling, construction, and validation of a simulator for dynamic processes, known as NPCTF.

Chapter 1 introduces the background of simulators and of the facility NPCTF. The motivations and objectives of this research are stated. The modeling solution and the specifications of the simulator are also described. At the end of this work, it can be concluded that all the objectives and the specifications have been fulfilled successfully.

Chapter 2 declares the configuration of the simulator, first with a conceptual framework, loop divisions, and loop connections. To implement the configuration, the modeling technique based on physical principles is adopted in developing models for NPCTF. The basic knowledge of thermal-hydraulic systems is then introduced as the foundation by which to form dynamic models of the Heater, Chiller, Pressurizer, HX-Tank, and Turbine. The configuration is the guideline for subsequent construction of the simulator diagram. The mathematical models are confirmed through simulations and validations against actual measurements from NPCTF.

Chapter 3 introduces the selected simulation environment and then describes the scheme for constructing the simulator based on the Simscape toolbox in Simulink of Matlab. The construction work is conducted successfully under the Simscape scheme.

Chapter 4 describes the entire construction process of the simulator diagram. The custom-defined blocks for specific functions of NPCTF are created by Simscape language. The diagram of the simulator is then converted to a functional simulator by using the blocks from the built-in library and the custom library. Through the V&V steps, the simulator diagram and the global model configuration can work to simulate the dynamic behaviours of NPCTF correctly.

Chapter 5 explains the V&V processes for the constructed simulator. In the processes, it demonstrates the functionalities of the simulator and reproduces the responses of NPCTF.

The validation process is the key step in confirming that the simulator can be accepted as the achievement of this research.

6.1 Conclusion

- The dynamic models based on physical principles can simulate the behaviours of the thermal-hydraulic processes of NPCTF. The models can also simulate other behaviours that NPCTF cannot easily produce, such as the cases in which variables exceed their operating ranges.

- The simulation environment in Matlab, with the Simscape toolbox, is sufficient for implementation of the simulator.

- The simulator has demonstrated the balances of energy, momentum, and mass in the dynamic processes of NPCTF. The details of the dynamic processes can be investigated in the simulator.

6.2 Summary of the simulator

The development of the simulator in the Simscape environment can be summarized in three phases:

1) Preparation of the component blocks by choosing built-in blocks of Simscape and creating custom-defined component blocks based on their dynamic models.
2) Construction of the simulator diagram under the configuration. The modeling solution of the physical network approach in Simscape is used to form the loops in the diagram within the hydraulic, pneumatic, and thermal domains.
3) Performance of simulations in the simulator. The simulation results are used to validate the models against the experimental data of NPCTF.

Through the development process of the simulator, some significant advantages of the simulator can be summarized as follows.

1) The simulator can be accelerated or decelerated by the setting in Simulink, or especially by changing the step size in the model global configuration.

2) There is no limitation in obtaining simulation data. The internal variables as well as the outputs of sensors in the models can be observed and obtained based on the constructed diagram or through redefining the blocks in the diagram.

3) The physical network approach of the toolbox Simscape makes it intuitive to reconstruct the diagram or modify parameters to observe more processes. Specially, some severe scenarios where the running parameters exceed the physical limits of NPCTF can be reached in the simulator.

4) The work of the simulator just depends on a computer. It can be duplicated for multiple users running simultaneously.

5) The simulator can work without worrying about cost, safety, or fading. Simulations can be conducted without considering safe ranges.

On the other hand, however, there are some drawbacks found along with the advantages.

a) The models have to be developed based on some assumptions and simplifications. The assumptions and simplifications will weaken the fidelity of the simulation models.

b) The toolbox Simscape runs the entire simulator system by solving equation matrix. This may produce mathematical issues such as non-converge or erroneous initial conditions. One should be careful when modifying the simulator.

c) Operations as well as initial conditions have to be set before executing the simulator. For thermal-hydraulic processes, this reduces the possibilities of simulating more behaviours.

d) Owing to the strong couplings of hydraulic systems, the simulations of some hydraulic processes are hard, especially during some validation processes. Their simulation conditions cannot be given as the same as those of experiments.

6.3 Summary of contributions

The major contributions of this research can be summarized as follows.

1) The simulator regarding NPCTF, as the simulation object, has been constructed by using the modeling technique based on physical principles. The characteristics of the dynamic processes exhibited in the simulator can be used to support related I&C designs and studies.

2) The developed dynamic models represent the underlying principles of the main components of NPCTF as well as some key mathematical relationships in NPP processes. Through the subsequent validations, the models have been confirmed to have the ability to reproduce the dynamic behaviours of NPCTF. The mathematical relations can further be used to aid in control design and investigation.

3) The Simscape codes of the main components have been developed to generate the custom-defined blocks to construct the simulator. The codes in textual files define the type, ports, variables, parameters, and equations of the component blocks. The codes are the result of implementing the dynamic models. With the generated blocks, the integrated simulator diagram can form the working mathematical relation matrix of the original system, and the simulator can simulate dynamic processes reflected by the underlying mathematical models.

4) Appropriate parameter settings of the component blocks have been tested and identified through simulations. The confirmed parameter settings represent the suitable parameters in simulating NPCTF. They can be applied in other types of simulation models developed by other modeling techniques or in other simulation environments.

5) Verification and validation for the simulator have been carried out to confirm the models. Through the verification steps, it is found that the simulator has met the designed specifications. The performance of the simulator is validated and accepted by comparisons between the simulations and the experiments. The V&V processes have confirmed that the purpose of this research has been fulfilled by the modeling, construction, and validation of the simulator.

6.4 Future work

The entire process for developing a simulator for NPCTF is presented in this thesis. Even though the simulator can conduct simulations for several dynamic processes in NPCTF, some subjects may further be investigated in future work.

- Distributed parameter models can be developed to simulate dynamic processes of heat transfer. Lumped parameter models have been used in this research. To reflect the distribution characteristics of temperatures, however, the distributed parameter model is a feasible option.

- A simulator in another supporting simulation environment with process models can be developed. Since the distinction between a training simulator and engineering simulator is not obvious, a simulator mixing these two types may be of more advantages for research.

- It is worthwhile to expand the simulation environment into real-time domain using, e.g. Real-Time Workshop® to support interaction with physical hardware. This will make it possible to conduct research on control system evaluation in a hardware-in-the-loop environment on this software simulator.

References

[1] J. M. Garrido, *Object Oriented Simulation - A Modeling and Programming Perspective*, Springer-Verlag US, Boston, MA, 2009

[2] D. Zanobetti, *Power station simulators*, ELSEVIER-Amsterdam-Oxford-New York-Tokyo, Amsterdam, New York, USA, 1989

[3] L. Ljung and T. Glad, *Modeling of Dynamic Systems*, Prentice Hall PTR, Englewood Cliffs, N.J., USA, 1994

[4] American Nuclear Society, *Nuclear power plant simulators for use in operator training and examination,* ANSI/ANS-3.5-2009, Sept. 2009

[5] International Atomic Energy Agency, *Use of control room simulators for training of nuclear power plant personnel*, IAEA-TECDOC-1411, Sept. 2004

[6] J. Jiang, J. P. Ma, A. Bari and D. J. Rankin, "A physical simulator in supporting of research & development for instrumentation and control systems in nuclear power plants," *9th International Topical Meeting on Nuclear Plant Instrumentation, Control, and Human-Machine Interface Technologies (NPIC & HMIT 2015)*, Charlotte, N.C., USA, Feb. 23-26, 2015

[7] J. C. Butcher, *Numerical Methods for Ordinary Differential Equations, 2nd Edition*, John Wiley, Chichester, England, 2008

[8] F. P. Incropera, D. P. Dewitt, T. L. Bergman, A. S. Lavine, *Fundamentals of Heat and Mass Transfer, 6th Edition*, John Wiley, Hoboken, N.J., USA, 2007

[9] K. M. Hangos, I. T. Cameron, *Process modeling and model analysis*, Academic Press, San Diego, California, USA, 2001

[10] P. J. Mohr, B. N. Taylor, D. B. Newell, "CODATA Recommended Values of the Fundamental Physical Constants: 2006," *Reviews of Modern Physics*, Vol. 80, April-June 2008

[11] GSE, *http://www.gses.com*, access date: Jan. 2015

[12] WSC, *https://www.ws-corp.com*, access date: Jan. 2015

[13] L-3 MAPPS, *http://www.mapps.l-3com.com*, access date: Jan. 2015

[14] CORYS, *http://www.corys.com*, access date: Jan. 2015

[15] APROS, *http://www.apros.fi/en/*, access date: Jan. 2015

[16] TRAX, *https://energy.traxintl.com/*, access date: Jan. 2015

[17] O. Merroun, A. Al Mers, M. A. Veloso, T. El Bardouni, B. El Bakkari, E. Chakir, "Experimental validation of the thermal-hydraulic code SACATRI," *Nuclear Engineering and Design*, 239 (2009) 2875-2884

[18] D. C. A. Lyria, C. Abla, L. Ahcene, B. S. Anis, "Thermal-hydraulic simulation of a radiant steam boiler using Relap5 computer code," *Computer and Chemical Engineering*, 52 (2013) 168-176

[19] A. P. Zhukavin, A. E. Kroshilin, and R. L. Fuks, "Development and Verification of Models Used in Training Simulators for Nuclear Power Stations," *ISSN 0040-6015, Thermal Engineering*, 2010, Vol. 57, No. 5, pp. 400-405

[20] H. Sato, H. Ohashi, S. Nakagawa, Y. Tachibana, and K. Kunitomi, "Validation and application of thermal hydraulic system code for analysis of helically coiled heat exchanger in high-temperature environment," *Journal of Nuclear Science and Technology*, Vol.51, Nos.11-12,1324-1335, 2014

[21] G. Bandini, P. Meloni, M. Polodori, C. Lombardo, "Validation of CATHARE V2.5 thermal-hydraulic code against full-scale PERSEO tests for decay heat removal in LWRS," *Nuclear Engineering and Design*, 241 (2011) 4662-4671

[22] Mathworks, *Simscape User's Guide R2014a*

[23] Mathworks, *Simscape Language Guide R2014a*

[24] Matlab, *http://www.mathworks.com*, access date: June 2015

[25] M. J. Thurgood, J. M. Kelly, T. E. Guidotti, R. J. Kohrt, K. R. Crowell, "COBRA/TRAC – A Thermal-Hydraulics Code for Transient Analysis of Nuclear Reactor Vessels and Primary Coolant Systems – Equations and Constitutive Models," NUREG/CR-3046, PNL-4385, Vol. 1 R4, 1983

[26] M. Kazimi and M. Massoud, *A condensed review of nuclear reactor thermal-hydraulic computer codes for two-phase flow analysis*, Massachusetts Institute of Technology, Cambridge, MA, 1980

[27] H. M. Hashemian, J. Jiang, "Nuclear plant temperature instrumentation," *Nuclear Engineering and Design*, 239 (2009) 3132-3141

[28] D. J. Rankin, J. Jiang, "A Hardware-in-the-Loop simulation platform for the verification and validation of safety control system," *IEEE Transactions on Nuclear Science*, vol. 58, No. 2, Apr. 2011

[29] B. V. Babu, *Process Plant Simulation*, Oxford University Press, New York, 2004

[30] International Atomic Energy Agency, *Guidelines for upgrade and modernization of NPP training simulators*, IAEA-TECDOC-1500, June 2006

[31] International Atomic Energy Agency, *Solution for cost effective assessment of software based instrumentation and control systems in nuclear power plants*, IAEA-TECDOC-1328, Dec. 2002

[32] International Atomic Energy Agency, *Means of evaluating and improving the effectiveness of training of nuclear power plant personnel*, IAEA-TECDOC-1358, Jul. 2003

[33] International Atomic Energy Agency, *Advanced nuclear plant design options to cope with external events*, IAEA-TECDOC-1487, Feb. 2006

[34] International Atomic Energy Agency, *Intercomparison and Validation of computer*

codes for thermalhydraulic safety analysis of heavy water reactors, IAEA-TECDOC-1395, Aug. 2004

[35] Z. Sui, B. G. Cui, Y. L. Ma, “The dynamic mathematic simulation model of steam generator for HTR-PM,” ICONE 19-43227, In *Proceedings of ICONE 19*, Osaka, Japan, Oct. 24-25, 2011

[36] S. D. Gennaro, B. Castillo, *Performance Study of the Control Systems in the presence of Faults and/or Reference Accidents in Pressurized Water Reactors of Evolutive Generation*, University of L’Aquila, Italy, Nov. 2011

[37] A. Hernandez-Solis, C. Ekberg, C. Demaziere, A. O. Jesen, U. Bredolt, “Uncertainty and sensitivity analyses as a validation tool for BWR bundle thermal-hydraulic predictions,” *Nuclear Engineering and Design*, 241 (2011) 3697-3706

[38] H. P. M. Gibcus, J. W. de Vries, P. F. A. de Leege, “Thermal hydraulic model validation for HOR mixed core fuel management,” *Nuclear Engineering and Design*, 180 (1998) 93-98

[39] K. Takamstsu, D. Tochio, S. Nakagawa, S. Takada, X. L. Yan, K. Sawa, N. Sakaba and K. Kunitomi, “Experiments and validation analyses of HTTR on loss of forced cooling under 30% reactor power,” *Journal of Nuclear Science and Technology*, Vol.51, Nos.11-12,1427-1443, 2014

[40] J. J. Jeong, K. S. Ha, B. D. Chung, W. J. Lee, “Development of a multi-dimensional thermal-hydraulic system code, MARS 1.3.1,” *Annals of Nuclear Energy*, 26 (1999) 1611-1642

[41] I. K. Park, J. R. Lee, S. W. Lee, H. Y. Yoon, J. J. Jeong, “An implicit code coupling of 1-D system code and 3-D in-house CFD code for multi-scaled simulations of nuclear reactor transients,” *Annals of Nuclear Energy*, 59 (2013) 80-91

[42] RELAP5/MOD3.3 Code Manual, NUREG/CR-5535/Rev 1 - Vol. I-IV,

Information Systems Laboratories, Inc., Rockville, Maryland, Dec. 2001

[43] A. L. Costa, P. Amelia L. Reis, C. Pereira, M. A. F. Veloso, A. Z. Mesquita, H. V. Soares, “Thermal hydraulic analysis of the IPR-R1 TRIGA research reactor using a RELAP5 model,” *Nuclear Engineering and Design*, 240 (2010) 1487-1494

[44] B. Siedel, V. Sartre, F. Lefevre, “Numerical investigation of the thermohydraulic behaviour of a complete loop heat pipe,” *Applied Thermal Engineering*, 61 (2013) 541-553

[45] K. J. Astrom, R. D. Bell, “Drum-boiler dynamics,” *Automatica*, 36 (2000) 363-378

[46] K. D. Kim, Rizwan-uddin, “A web-based nuclear simulator using RELAP5 and LabVIEW,” *Nuclear Engineering and Design*, 237 (2007) 1185-1194

[47] A. Prosek, B. Mavko, “Animation model of Krsko nuclear power plant for RELAP5 calculations,” *Nuclear Engineering and Design*, 241 (2011) 1034-1046

[48] A. Prosek, B. Mavko, “RELAP5/MOD3.3 Code Validation with Plant Abnormal Event,” Hindawi Publishing Corparation, *Science and Technology of Nuclear Installations*, doi: 10.1155/2008/745178

[49] A. Prosek, F. D’Auria, D. J. Richards, B. Mavko, “Quantitative assessment of thermal-hydraulic codes used for heavy water reactor calculations,” *Nuclear Engineering and Design*, 236 (2006) 295-308

[50] Argonne National Laboratory, *Summary of RELAP5 Assessments Performed in Relation to Conversion Analysis of Research Reactors*, ANL/GTRI/TM-14-3, Apr. 2014

[51] M. Rehman and S. A. Pedersen, “Validation of simulation models,” *Journal of Experimental & Theoretical Artificial Intelligence*, Vol. 24, No. 3, 351-363, September 2013

[52] I. Hafner, M. Robler, B. Heizl, A. Korner, M. Landsiedl, F. Breitenecker, "Investigating communication and step-size behaviour for co-simulation of hybrid physical systems," *Journal of Computational Science*, 5 (2014) 427-438

[53] M. Lin, D. Hou, P. F. Liu, Z. W. Yang, Y. H. Yang, "Main control system verification and validation of NPP digital I&C system based on engineering simulator," *Nuclear Engineering and Design*, 240 (2010) 1887-1896

[54] K. Velten, *Mathematical Modeling and Simulation – Introduction for Scientists and Engineers*, Wiley-VCH Verlag GmbH & Co. KGaA, Weinheim, Germany, 2009

[55] J. P. Holman, *Heat Transfer, 8th edition*, McGraw-Hill Inc, 1997

[56] L. Ljung, *System Identification: Theory for the User*, Prentice Hall PTR, Upper Saddle River, NJ, 1999

[57] *http://en.wikipedia.org/wiki/Convective_heat_transfer*, *http://en.wikipedia.org/wiki/Boundary_layer*, *http://en.wikipedia.org/wiki/Pressure*, *http://en.wikipedia.org/wiki/Ideal_gas*, *https://en.wikipedia.org/wiki/Real-time_simulation*, access date: June 2015

[58] K. Frezza, M. K. Jaby, "Assessment of simulation advantages on nuclear DCS projects," *9th International Topical Meeting on Nuclear Plant Instrumentation, Control, and Human-Machine Interface Technologies (NPIC & HMIT 2015)*, Charlotte, N.C., USA, Feb. 23-26, 2015

[59] B. Roffel and B. Betlem, *Process Dynamics and Control - Modeling for Control and Prediction*, John Wiley & Sons, Chichester, England, 2006

[60] B. G. Cui and J. Jiang, "Development of dynamic models for thermal-hydraulic processes based on Simscape Toolbox," *9th International Topical Meeting on Nuclear Plant Instrumentation, Control, and Human-Machine Interface Technologies (NPIC & HMIT 2015)*, Charlotte, N.C., USA, Feb. 23-26, 2015

Appendices

Appendix A: Introduction to Simscape

Simscape is a toolbox in Matlab Simulink. It provides a single environment for modeling and simulating physical systems. Through assembling the blocks from its built-in fundamental library and the developed custom-defined library into the diagrams of the models, the simulation system can perform simulations by employing the definite solver to solve the integrated equation matrix. Since Simscape components use physical connections, the formed model diagrams can match the structure of the modeling system.

The main features and the useful information for building the models of NPCTF are introduced in this appendix. More documents about Simscape and Simulink can reference the official website of mathworks.

A.1 Characteristics of Simscape

Simscape is of the key features as:

- The physical network approach for modeling and simulating hydraulic, thermal, pneumatic, electrical, and other multidomain physical systems.
- The built-in libraries of physical modeling blocks, and mathematical elements for developing custom-defined components.
- Matlab based Simscape language enabling text-based programming of physical modeling components, domains, and libraries.
- Physical units for parameters and variables, with all unit conversions handled automatically.
- Employing other Simulink blocks except the Simscape blocks as the extending block source.
- Support for C-code generation.

A.2 Relationship of Simulink and Simscape

Simulink is a block diagram environment for multi-domain simulation and model-based design. It supports simulation, automatic code generation, and continuous test and verification of embedded systems. Simulink provides a graphical editor, customizable block libraries, and solvers for modeling and simulating dynamic systems. It is integrated with Matlab. Then it can incorporate Matlab algorithms into models and export simulation results to Matlab for further analysis. Some key features of Simulink are illustrated as:

- Graphical editor for building and managing hierarchical block diagrams.
- Libraries of predefined blocks for modeling continuous-time and discrete-time systems.
- Simulation engine with fixed-step and variable-step ODE solvers.
- Scope and data displays for viewing simulation results.
- Incorporation with Matlab workspace and sharing simulation data.

As it is mentioned, Simscape is a toolbox in Simulink. Therefore, all the features of Simulink can be used in Simscape modeling process. As a specific toolbox, moreover, Simscape supports physical modeling with the relevant features rather than other toolbox. The communications between the built-in blocks of Simscape and the fundamental blocks of Simulink need the PS-Simulink blocks as the converters.

A.3 Basic concepts of Simscape

1) Physical System Modeling

The modeling technique in Simscape is employing a physical network approach to build model systems. Different from the standard Simulink modeling approach, the physical network approach is particularly suited to simulating systems that consist of real physical components. The components corresponding to physical elements, such as pumps, valves, pipes, tanks, and other component blocks, are joined by lines corresponding to the physical connections in the real system. This modeling technique allows describing the physical structure of a system rather than the underlying mathematics. From the model diagram, Simscape automatically constructs the differential algebraic equations (DAEs) that characterize the behaviour of the system. The variables for the components in the different physical domains are solved simultaneously.

2) Physical Domains and Variable Types

In Simscape, the common physical domains are predefined. The domain definitions of hydraulic, thermal, and pneumatic domain are shown as the follows.

```
domain hydraulic
% Hydraulic Domain
% Copyright 2005-2008 The MathWorks, Inc.

  parameters
    density       = { 850    , 'kg/m^3' }; % Fluid density
    viscosity_kin = { 18e-6 , 'm^2/s'  }; % Kinematic viscosity
    bulk         = { 0.8e9 , 'Pa'      };% Bulk modulus at atm.
pressure and no gas
    alpha        = { 0.005 , '1'       }; % Relative amount of
trapped air
  end

  variables
    p = { 0 , 'Pa' }; % pressure
  end

  variables(Balancing = true)
    q = { 0 , 'm^3/s' }; % flow rate
  end
```

(a) hydraulic

```
domain thermal
% Thermal domain
% Copyright 2005-2008 The MathWorks, Inc.

  variables
    T = { 293.15 , 'K' }; % temperature
  end

  variables(Balancing = true)
    Q = { 0 , 'J/s' };  % heat power
  end

end
```

(b) thermal

```
domain pneumatic
% Pneumatic 1-D Flow Domain
% Copyright 2008-2009 The MathWorks, Inc.

  parameters
    gam = { 1.4, '1' };                 % Ratio of specific heats
    c_p = { 1005 , 'J/kg/K' };          % Specific heat at constant
pressure
    c_v = { 717.86 , 'J/kg/K' };        % Specific heat at constant
volume
    R   = { 287.05, 'J/kg/K' };         % Specific gas constant
    viscosity = { 18.21e-6, 'Pa*s' }; % Dynamic viscosity
    kin_viscosity = { 15.11e-6, 'm^2/s'};    % Kinematic viscosity
    Pa   = { 101325, 'Pa' };            % Ambient pressure
    Ta   = { 293.15, 'K' };             % Ambient temperature
  end

  variables
    p = { 1e5 , 'Pa' }; % pressure
    T = { 300 , 'K' };  % temperature
  end

  variables(Balancing = true)
    G = { 0 , 'kg/s' };  % flow rate
    Q = { 0 , 'J/s' };   % heat power
  end
```

(c) pneumatic

Figure A-1 Domain definitions in Simscape

Two types of variables are supported by the physical network approach:

- Through - Variables that are measured with a gauge connected in series to an element.

- Across - Variables that are measured with a gauge connected in parallel to an element.

These two corresponding variables in the foundation physical domains are shown in Table A-1.

Table A-1 Predefined physical domains

Physical Domain	Across Variable	Through Variable
Hydraulic	Pressure	Flow rate
Thermal	Temperature	Heat flow
Pneumatic	Pressure and temperature	Mass flow rate and heat flow
Electrical	Voltage	Current
Magnetic	Magneto motive force	Flux
Mechanical (rotational, translational)	Velocity	Torque or Force
Thermal Liquid	Pressure and temperature	Mass flow rate and thermal flux

3) Simscape Built-in Block Library

Simscape block library contains the block products of physical modeling family. The contained libraries are owed to their installations. The complete package is described in Table A-2.

Table A-2 Built-in block library

Name	Modeling Domain
Foundation Library	Contains basic blocks of electrical, hydraulic, magnetic, mechanical, physical signals, pneumatic, thermal, and thermal liquid.
SimDriveLine	Mechanical driveline blocks
SimElelctronics	Electrical circuit blocks
SimHydraulics	Hydraulic system blocks
SimMechanics	Mechanical system blocks

SimPowerSystems	Electrical power system blocks
Utilities	Contain the blocks of connection port, Simulink-Physical Simulink converter, solver configuration, etc.

4) Connecting Ports

Simscape blocks contain the two types of ports: physical conserving ports and physical signal ports.

1) Physical conserving ports (▪). These non-directional ports represent physical connections and relate physical variables based on the physical network approach. The conserving ports are connected only in the same types. The variables of the port (Across variables) have the same values on the directly connecting ports.

2) Physical signal ports (▷). These unidirectional ports are used for transferring internal signals. Unlike Simulink signals, which are essentially unitless, these ports carry physical signals which can have units. The units can be specified along with the parameter values in the block dialogs, and Simscape software performs the necessary unit conversion operations when solving the physical network.

A.4 Custom-defined components in Simscape

There are two types of component models in Simscape: a) Behavioral and b) Composite. Behavioral model is described by a system of mathematical equations and implemented based on its physical behaviours. Composite model is constructed out of other blocks and connected in a certain way.

Simscape language is the programming language to create new behavioral and composite models when the design requirements of users are not satisfied by the standard libraries provided with Simscape. It is a dedicated textual language for modeling physical systems. By using the physical foundation domains, the user- building components are compatible with the standard Simscape components and can be shared through the custom-defined block library.

To create a new custom-defined component, a component file shall be written. In this file, it must begin with the *component* keyword, followed by the component name, and be terminated by the *end* keyword. A component file typically contains the following sections:

- Declaration. It contains all the member class declarations for the component, such as parameters, variables, nodes, inputs, and outputs. Each member class declaration is a separate declaration block, which begins with the appropriate keyword (corresponding to the member class) and is terminated by the *end* keyword.

- Setup. It prepares the component for simulation. The body of the *setup* function can contain assignment statements, *if* and *error* statements, and *across* and *through* functions. The *setup* function is executed once for each component instance during model compilation. It takes no arguments and returns no arguments.

- Structure. It declares the component connections for composite models.

- Equation. It declares the component equations for behavioral models. It is executed throughout the simulation. The purpose of the equation section is to establish the mathematical relationships among the variables, parameters, inputs, outputs, time and the time derivatives of each of these entities in a component.

After the textual component files are created, they are converted into a component library of blocks by using the procedure:

1) Organizing all the Simscape files into package directories. The package hierarchy determines the resulting library structure. The package directory name must begin with a +character.

2) Optional source protection if sharing the models without disclosing the component or domain source.

3) Building the custom-defined block library by using the command *ssc_build* at the top-level package directory in the command window of Matlab. A separate custom-

defined Simscape block library will be generated in the parent directory of the top-level package.

A.5 Construction of simulation models in Simscape

There are some essential modeling techniques that need to be followed in building model system:

- The model diagram is built by using a combination of blocks from the Simscape built-in library and the custom-defined library. The model diagram represents the network of the simulation object, in which the functional elements interact with each other by exchanging information through their ports.

- Each Simscape diagram or each topologically distinct physical network in a diagram must contain a Solver Configuration block from the Simscape Utilities library, which is as the local solver configuration.

- Within a physical network, each domain or each topologically distinct circuit must contain at least one reference block of the appropriate type. More than one reference block may be used within a circuit to define multiple connections to the domain reference.

- Hydraulic global parameters in a hydraulic circuit, such as fluid density and fluid kinematic viscosity, are defined in the definition of the hydraulic domain. If need to specify the working fluid, one can attach a Hydraulic Fluid block to each topologically distinct hydraulic circuit.

- The same as hydraulic domain, if there is no Gas Properties block, pneumatic elements in a pneumatic circuit use the default gas properties with the ambient conditions of 101325 Pa and 20 degrees Celsius. By attaching a Gas Properties block to each topologically distinct pneumatic circuit, one can change gas properties and ambient conditions in this block.

- To connect regular Simulink blocks to the physical network diagram, the converter blocks are needed.

The following rules apply to using conserving ports:

- The different types of physical conserving ports used in Simscape block diagrams, such as hydraulic, pneumatic, electrical, magnetic, thermal, mechanical translational, and mechanical rotational, contain their specific through and across variables. This is listed in the table I.

- The conserving ports are connected only to the same type ports.

- The connection lines between the conserving ports are non-directional lines. The lines carry across variables and through variables rather than signals. The physical lines cannot be connected to Simulink ports or to physical signal ports.

- Two directly connected conserving ports have the same values for all their across variables.

- Physical connection lines can be branched. In this case, the same across variables are shared, and through variables are divided among the multiple components connected by the branches. The division of the through variable is determined by the system dynamics. For each through variable, the sum of all its values flowing into a branch point equals the sum of all its values flowing out.

The following rules apply to physical signal ports:

- Physical signal ports are connected to other physical signal ports with regular connection lines, which is similar to Simulink signal connections. These connection lines carry physical signals between Simscape blocks.

- Physical signal ports in Simscape can connect to Simulink ports through special converter blocks. The Simulink-PS Converter block is used to connect Simulink outports to Physical Signal inports. The PS-Simulink Converter block is used to connect Physical Signal outports to Simulink inports.

- Physical signals can contain units associated with them. The converter blocks between Simscape blocks and Simulink locks can specify the desired units. All unit conversions are handled automatically.

A.6 Choice of solvers

The job of a solver is to apply numerical methods to solve a set of ordinary differential equations summarized from the models. In the process of solving the initial value problems, the solvers also satisfy the specified accuracy requirements. The following tables summarize the types of solvers in the Simulink library. It is noted that every solver can perform on the models that contain algebraic loops.

Table A-3 Solver type

		Continuous	**Variable-order**
Fixed-step	Explicit	ode1, ode2, ode3, ode4, ode5, ode8	Not Applicable
	Implicit	ode14x	Not Applicable
Variable-step	Explicit	ode45,ode23, ode113	ode113
	Implicit	ode15s, ode23s, ode23t, ode23tb	ode15s

Table A-4 Explicit continuous fixed-step solvers

Solver	**Integration Technique**	**Order of Accuracy**
ode1	Euler's Method	First
ode2	Heun's Method	Second
ode3	Bogacki-Shampine Formula	Third
ode4	Fourth-Order Runge-Kutta (RK4) Formula	Fourth
ode5	Dormand-Prince (RK5) Formula	Fifth
ode8	Dormand-Prince RK8(7) Formula	Eighth

Table A-5 Implicit continuous fixed-step solvers

ODE Solver	Method	Order	Order of Accuracy
`ode14x`	Combination of Newton's method and extrapolation	X	Corresponding to the order selection

Table A-6 Explicit continuous variable-step solvers

ODE Solver	Method	One-Step Method	Multistep Method	Order of Accuracy
`ode45`	Runge-Kutta, Dormand-Prince (4,5) pair	X		Medium
`ode23`	Runge-Kutta (2,3) pair of Bogacki & Shampine	X		Low
`ode113`	PECE Implementation of Adams-Bashforth-Moutlon		X	Variable, Low to High

Table A-7 Implicit continuous variable-step solvers

ODE Solver	Method	One-Step Method	Multistep Method	Solver Reset Method	Max. Order	Order of Accuracy
`ode15s`	Numerical Differentiation Formulas (NDFs)		X	X	X	Variable, Low to Medium
`ode23s`	Second-order, modified Rosenbrock formula	X				Low
`ode23t`	Trapezoidal rule using a "free" interpolant	X		X		Low
`ode23tb`	TR-BDF2	X		X		Low

Both fixed-step and variable-step solvers compute the next simulation time as the sum of the current simulation time. Step size is known time as a quantity. With a fixed-step solver, the step size remains constant throughout the simulation. In contrast, with a variable-step solver, the step size can vary from step to step, depending on the model dynamics. The choice between the two types depends on how to deploy the models and the model dynamics. A variable-step solver might shorten the simulation time of a model significantly by adjusting the step size dynamically and thus reducing the number of steps. Whereas a fixed-step solver must use a single step size throughout the simulation based upon the accuracy requirements.

Implicit solvers are designed specifically for solving stiff problems whereas explicit solvers are used to solve non-stiff problems. A general definition of a stiff system is a system that has extremely different time scales. Compared to the explicit solvers, the implicit solvers provide greater stability for oscillatory behaviour, and generate the Jacobian matrix and solve the set of algebraic equations at every time step using a Newton-like method. To reduce this extra expensive cost, the implicit solvers offer a Solver Jacobian method parameter that allows users to improve the simulation performance of implicit solvers.

For physical models such as Simscape models, implicit solvers such as ode14x, ode 23s, and ode 15s, are recommended, since the implicit solvers require fewer time steps and fewer input variables than explicit solvers such as ode45, ode113, and ode1.

Appendix B: Complete Simscape Code of Custom-defined Components

B.1 Code of the Heater block

```
component pctf_heater < foundation.hydraulic.branch
% Heater
% This block represents the model of the Heater.
%
% I : connecting to current signal which controls the power heater, A\n
% Tamb : the input of ambient temperature, K \n
% Gin_fortest : just for test mode, the flow rate = the input, m^3/s \n
% mode : test mode switch, 0: normal; 1: test mode. \n
% Vol : the inner volume of heater, m^3 \n
% Gv : water flow rate, m^3/s \n
% Tshh : the temperature of metal sheath, K \n
% Qw : heat transfer flux between water and sheath, J/s \n
% Ql : heat lost to ambient, J/s \n
% dp : pressure difference between inlet and outlet, Pa \n
% denw : density of water, kg/m^3 \n
% Pr : Prandtl number \n
% vel : velocity of water, m/s \n
% Re : Reynolds number \n
% Nu : Nusselt number \n
% coh : convective heat transfer coefficient, W/(m^2*K) \n
%
% Nodes Tin,Tout : conserving thermal ports, connect water temperatures
%
% Connections A and B are conserving hydraulic ports associated
% with the block inlet and outlet, respectively. The block positive
% direction is from port A to port B. This means that the flow rate is
% positive if fluid flows from A to B, and the pressure loss is
% determined as dp = p_A - p_B.
%
% Copyright 2014-2015 UWO.ECE.CIES

 nodes
    Tout= foundation.thermal.thermal; % Tout:left
    Tin = foundation.thermal.thermal; % Tin:left
 end

 inputs
    Ie = { 0, 'A' }; % I
    Tamb = { 293.15, 'K' };     % Tamb:left
    Gin = { 0.108 , 'm^3/s' }; % Gin_fortest
    mode = { 0, '1'}; %mode
 end

 outputs
    Vol = { 0.5562e-3, 'm^3' }; %Vol
    Gv = { 1, 'm^3/s' }; % Gv
    Tshh = { 293.15, 'K'};%Tshh
    Qw ={ 0,  'J/s'};  %Qw
    Ql ={ 0,  'J/s'};  %Ql
    dp ={ 0,  'Pa'};   %dp
```

```
    denw = { 1.e+3, 'kg/m^3' };%denw
    Pr ={ 1, '1' } ;            %Pr
    vel ={ 0, 'm/s' };          %vel
    Re ={ 0, '1' } ;            %Re
    Nu ={ 3.66, '1' } ;         %Nu
    coh ={100e+3, 'W/(m^2*K)' };%coh
 end

 variables
    Ty = { 293.15, 'K' };       % outlet temperature
    Tmetal = { 293.15, 'K' };   % sheath temperature
    Qe = { 0 , 'J/s' };         % thermal energy converted from electric
    Qs = { 0, 'J/s' }; % recerved
    Ts = { 0, 'K' };   % reserved
    resistance = { 1.e+7, 'Pa/(m^3/s)' }; % resistance
    Cpin = { 4.18e+3, 'J/(kg*K)' };    % specific heat of inlet water
    Cpout= { 4.18e+3, 'J/(kg*K)' };    % specific heat of outlet water
    Ashh ={ 0.1, 'm^2'};        % surface area of sheath
    Dhyd ={ 14.097, 'mm' };     % hydraulic diameter
    lambda ={ 0.6, 'W/(m*K)' } ;% thermal conductivity
    Nu_l ={ 3.66, '1' } ;%Nu laminar
    Nu_t ={ 3.66, '1' } ;%Nu turbulent
 end

 parameters
    resisdefault = { 7.5e+7, 'Pa/(m^3/s)' }; % hydraulic resistance
    ita = { 0.97, '1'};              % efficiency of energy conversion
    Ue = { 240, 'V' };               % voltage of electrical source
    Dhtr_o = { 2.5, 'in' };          % outer diameter of heater
    Dhtr = { 1.5, 'in' };            % inner diameter of heater
    Dshh = { 0.315, 'in' };          % diameter of sheath
    Hhtr = { 20, 'in' };             % height of heater
    Him = { 18, 'in' };              % height immersed
    mshh = {4.0, 'kg'};              % equivalent mass of sheath
    Cpshh = { 460, 'J/(kg*K)' };     % specific heat of sheath
    epsi = { 0.99, '1' }; % coefficient of heat lost to ambient
    corr_l ={[0.664  0.5 0.33], '1'}; % Nusselt number correlation
coefficients of laminar regime
    corr_t ={[1.56 0.85 0.33], '1'};  % Nusselt number correlation
coefficients of turbulent regime
    Re_l ={ 1000, '1' } ;      % laminar flow upper margin
    Re_t ={ 2300, '1' } ;      % turbulent flow lower margin
    Tao_e ={ 10, 's' };        % time constant of energy conversion
    Ie0 = { 0, 'A' };          % initial current into heater
    T0 = { 293.15 , 'K' };     % initial temperature of heater
  end
  parameters(Access =private)
    t_unit = { 1 , 's' };      % uniform units
    Kcoh ={ 1, '1' };          % correction coefficient to heat transfer
coefficient
    Sigma0 ={ 5.67e-8, 'W/(m^2*K^4)' }; % Stefan-Boltzmann constant
    T_y={[273.16:10:373.16],'K'};
    lambda_y={[0.5513 0.5745 0.5989 0.6176 0.6338 0.6478 0.6594 0.6676
0.6745 0.6798 0.6804],'W/(m*K)' };
    Cp_y={[4.212 4.191 4.183 4.174 4.174 4.174 4.178 4.178 4.195 4.208
4.220],'J/(g*K)'};
```

```
    miu_y={[1.7921e-3 1.3077e-3 1.0050e-3 0.8007e-3 0.6560e-3 0.5494e-3
0.4688e-3 0.4061e-3 0.3565e-3 0.3165e-3 0.2838e-3],'Pa*s'};
    denw_y={[999.9 999.7 998.2 995.7 992.2 988.1 983.2 977.8 971.8
965.3 958.4], 'kg/m^3'};
  end

  function setup
    Tmetal = T0;  % ensure correct initial conditions
    Ty = T0;
    Qe = ita*Ue*Ie0; % avoid big difference at first.
  end

  branches
    Qs : Tin.Q -> Tout.Q;
  end

  equations
     Ts == Tin.T - Tout.T;
     Vol== pi/4*Dhtr^2*Hhtr-pi/4*Dshh^2*(2*Him+Dshh); % inner volume
     resistance == resisdefault ; % hydraulic resistance
     p == resistance * q;
     Gv == if (mode==0),q else Gin end; % volumetric flow rate, m^3/s
     Cpin ==tablelookup(T_y, Cp_y, Tin.T, interpolation=linear,
extrapolation=nearest);
     Cpout==tablelookup(T_y, Cp_y, Tout.T,interpolation=linear,
extrapolation=nearest );
denw==(tablelookup(T_y,denw_y,Tin.T)+tablelookup(T_y,denw_y,Tout.T))/2;
     lambda==tablelookup(T_y, lambda_y, (Tin.T+Tout.T)/2,
interpolation=linear, extrapolation=linear);
     Pr ==((A.viscosity_kin+B.viscosity_kin)/2)/lambda*denw*
tablelookup(T_y,Cp_y,(Tin.T+Tout.T)/2,interpolation=linear,extrapolatio
n=nearest); % Pr number
     Dhyd ==Dhtr-3*Dshh ;
     vel ==Gv /(pi/4*Dhyd^2); % velocity
     Re ==vel/((A.viscosity_kin+B.viscosity_kin)/2)*Dhyd ; %Re number
     Nu_l ==corr_l(1)*power(Re,corr_l(2))*power(Pr,corr_l(3)) ;
     Nu_t ==corr_t(1)*power(Re,corr_t(2))*power(Pr,corr_t(3)) ;
     if ( Re<=Re_l )
         Nu ==Nu_l ;
     elseif ( Re>=Re_t )
         Nu ==Nu_t ;
     else %transition
         Nu ==Nu_l+(Nu_t-Nu_l)*(Re-Re_l)/(Re_t-Re_l) ;
     end
     coh ==Kcoh* lambda*Nu/Dhyd ; %convective heat transfer coefficient

     if (Ie>=0)
         Qe.der ==(ita*Ue*Ie-Qe)/Tao_e;
     else
         Qe.der == -Qe/t_unit; % Qe==0
     end

     Ashh ==pi* Dshh*(2*Him+Dshh) ;
     Qw ==(Tmetal-Ty)*coh *Ashh; % heat exchange between sheath and
water
```

```
     Ql == epsi* Sigma0*(pi*Dhtr_o*Hhtr + 2*pi/4*Dhtr_o^2)*(Ty^4 -
Tamb^4);
     Tmetal.der == (Qe-Qw)/(mshh*Cpshh);

     if (Gv>=0)
       Ty.der==(Gv*A.density*Cpin*Tin.T-Gv*B.density*Cpout*Ty+Qw-
   Ql)/(denw*Vol*Cpout);
        Tout.T == Ty; % K
     else
       Ty.der==(-Gv*B.density*Cpout*Tout.T+Gv*A.density*Cpin*Ty+Qw-
   Ql)/(denw*Vol*Cpin);
        Tin.T == Ty; % K
     end
     dp ==p ; % difference pressure
     Tshh ==Tmetal;
  end

end
```

B.2 Code of the Chiller block

```
component pctf_chiller < foundation.hydraulic.branch
% Chiller
% This block represents the model of the Chiller.
%
% Tamb : the input of ambient temperature, K \n
% Gin_fortest : just for test mode, the flow rate = the input, m^3/s \n
% mode : test mode switch, 0: normal; 1: test mode. \n
% Gv : water flow rate of primary side, m^3/s \n
% Tclr : the temperature of metal lamellas, K \n
% Qw : heat transfer flux of primary side, J/s \n
% Ql : heat lost to ambient, J/s \n
% dp : pressure difference of inlet and outlet of primary side, Pa \n
% denw : density of water, kg/m^3 \n
% Pr : Prandtl number \n
% vel : velocity of water, m/s \n
% Re : Reynolds number \n
% Nu : Nusselt number \n
% coh : convective heat transfer coefficient, W/(m^2*K) \n
%
% Nodes Tin, Tout : thermal conserving ports,  used to connect water
% temperatures of primary water loop.
%
% Connections A and B: conserving hydraulic ports, used to connect to
% primary water loop as inlet and outlet. The flow rate is positive
% if fluid flows from A to B.
%
% Nodes A2, B2, Tin2, Tout2: used to connect to those in secondary
% side, with the same function as primary side.
%
% Copyright 2014-2015 UWO.ECE.CIES

 nodes
    Tin = foundation.thermal.thermal; % Tin:right
    Tout= foundation.thermal.thermal; % Tout:right
    A2  = foundation.hydraulic.hydraulic; % A2:right
```

```
   B2  = foundation.hydraulic.hydraulic; % B2:right
   Tin2 = foundation.thermal.thermal; % Tin2:right
   Tout2= foundation.thermal.thermal; % Tout2:right
end

inputs
   Tamb = { 293.15, 'K' }; % Tamb:right
   Gin  = { 1 , 'm^3/s' }; % Gin_fortest:right
   Gin2 = { 1 , 'm^3/s' }; % Gin2_fortest:right
end

outputs
   Gv = { 1, 'm^3/s' };    % Gv:left
   Gv2 = { 1, 'm^3/s' };   % Gv2:left
   Tclr = { 293.15 , 'K'};% Tclr:left
   Qw ={ 0,  'J/s'};    % Qw:left
   Ql ={ 0,  'J/s'};    % Ql:left
   dp ={ 0,  'Pa'};     % dp:left
   denw = { 1.e+3, 'kg/m^3' }; % denw:left
   Pr ={ 1, '1' } ;            % Pr:left
   vel ={ 0, 'm/s' };          % vel:left
   Re ={ 0, '1' } ;            % Re:left
   Nu ={ 3.66, '1' } ;         % Nu:left
   coh ={1.e+6, 'W/(m^2*K)' }; % coh:left
   Pr2 ={ 1, '1' } ;           % Pr2:left
   vel2 ={ 0, 'm/s' };         % vel2:left
   Re2 ={ 0, '1' } ;           % Re2:left
   Nu2 ={ 3.66, '1' } ;        % Nu2:left
   coh2 ={1.e+6 'W/(m^2*K)' }; % coh2:left
 end

 variables
   Ty = { 293.15 , 'K' };      % outlet temperature
   Tmetal = { 293.15 , 'K' };  % metal temperature
   Qs = { 0, 'J/s' };  %reserved
   Ts = { 0, 'K' };    %reserved
   Qs2 = { 0, 'J/s' }; %reserved
   Ts2 = { 0, 'K' };   %reserved
   resistance = { 1.e+7, 'Pa/(m^3/s)' }; % resistance
   Cpin = { 4.18e+3, 'J/(kg*K)' };   % specific heat of inlet water
   Cpout= { 4.18e+3, 'J/(kg*K)' };   % specific heat of outlet water
   lambda ={ 0.6, 'W/(m*K)' } ; % thermal conductivity
   Nu_l ={ 3.66, '1' } ; %Nu laminar
   Nu_t ={ 3.66, '1' } ; %Nu turbulent

   q2 = { 1, 'm^3/s' }; % flow rate of secondary side
   p2 = { 0 , 'Pa' };   % pressure of secondary side
   denw2 = { 1.e+3, 'kg/m^3' };
   Ty2 = { 293.15 , 'K' };
   Qw2 ={ 0, 'J/s'}; % Qw2:left
   Cpin2 = { 4.18e+3, 'J/(kg*K)' };   % specific heat of inlet water
   Cpout2= { 4.18e+3, 'J/(kg*K)' };   % specific heat of outlet water
   lambda2 ={ 0.6, 'W/(m*K)' } ;      % thermal conductivity
   Nu_l2 ={ 3.66, '1' } ; % Nu laminar
   Nu_t2 ={ 3.66, '1' } ; % Nu turbulent
 end
```

```
  parameters
    testmode ={0,'1'} ; % test mode switch, 0: normal; 1: test mode.
    resisdefault = { 9.5e+7, 'Pa/(m^3/s)' }; % hydraulic resistance of
primary side
    resisdefault2 = { 9.e+7, 'Pa/(m^3/s)' }; % hydraulic resistance of
secondary side
    Vol = { 0.5e-3, 'm^3' };        % inner volume
    Ainner = { 3.e-1, 'm^2'};    % inner surface area
    Aouter = { 0.5e-1, 'm^2'};  % outer surface area
    Dclr ={ 35, 'mm'}; % hydraulic diameter
    Lclr ={ 200, 'mm'};% length of flow path
    mclr = {5.0, 'kg'};                 % mass of lamella
    Cpclr = { 500, 'J/(kg*K)' };     % specific heat of lamella
    epsi = { 0.99, '1' }; % coefficient of heat lost to ambient
    corr_l ={[0.664  0.5 0.33], '1'}; % Nusselt number correlation
coefficients of laminar regime
    corr_t ={[0.316 0.8 0.33], '1'};  % Nusselt number correlation
coefficients of turbulent regime
    Re_l ={ 2000, '1' } ; % laminar flow upper margin
    Re_t ={ 4000, '1' } ; % turbulent flow lower margin
    T0 = { 293.15 , 'K' };% initial temperature of chiller
  end
  parameters(Access =private)
    t_unit = { 1 , 's' };  % uniform units
    Kcoh ={ 1, '1' };  % correction coefficient to heat transfer
coefficient of 1st side
    Kcoh2 ={ 1, '1' }; % correction coefficient to heat transfer
coefficient of 2nd side
    Sigma0 ={ 5.67e-8, 'W/(m^2*K^4)' }; % Stefan-Boltzmann constant
    T_y={[273.16:10:373.16],'K'};
    lambda_y={[0.5513 0.5745 0.5989 0.6176 0.6338 0.6478 0.6594 0.6676
0.6745 0.6798 0.6804],'W/(m*K)' };
    Cp_y={[4.212 4.191 4.183 4.174 4.174 4.174 4.178 4.178 4.195 4.208
4.220],'J/(g*K)'};
    miu_y={[1.7921e-3 1.3077e-3 1.0050e-3 0.8007e-3 0.6560e-3 0.5494e-3
0.4688e-3 0.4061e-3 0.3565e-3 0.3165e-3 0.2838e-3],'Pa*s'};
    denw_y={[999.9 999.7 998.2 995.7 992.2 988.1 983.2 977.8 971.8
965.3 958.4], 'kg/m^3'};
  end

  function setup
    Tmetal = T0; % ensure correct initial conditions
    Ty = T0;
    Ty2 = T0;
  end

  branches
    Qs : Tin.Q -> Tout.Q;
    Qs2 : Tin2.Q -> Tout2.Q;
    q2 : A2.q -> B2.q;
  end

  equations
     Ts == Tin.T - Tout.T;
     Ts2 == Tin2.T - Tout2.T;
```

```
    p2 == A2.p - B2.p;
    p2 == resisdefault2 * q2;
    resistance == resisdefault ;
    p == resistance * q;
    Gv ==q*(1-testmode)+Gin*testmode;
    Gv2==q2*(1-testmode)+Gin2*testmode;
    Cpin ==tablelookup(T_y, Cp_y, Tin.T, interpolation=linear,
extrapolation=nearest);
    Cpout==tablelookup(T_y, Cp_y, Tout.T,interpolation=linear,
extrapolation=nearest );
     denw==(tablelookup(T_y,denw_y,Tin.T)+tablelookup(T_y,denw_y,Tout.
T))/2;
    lambda==tablelookup(T_y, lambda_y, (Tin.T+Tout.T)/2,
interpolation=linear, extrapolation=linear);
    Pr ==((A.viscosity_kin+B.viscosity_kin)/2)/lambda*denw*
tablelookup(T_y,Cp_y,(Tin.T+Tout.T)/2,interpolation=linear,extrapolatio
n=nearest); % Pr number
    vel ==Gv /(pi/4 * Dclr^2); % velocity
    Re ==vel/((A.viscosity_kin+B.viscosity_kin)/2)*Dclr ; %Re number
    Nu_l ==corr_l(1)*power(Re,corr_l(2))*power(Pr,corr_l(3)) ;
    Nu_t ==corr_t(1)*power(Re,corr_t(2))*power(Pr,corr_t(3)) ;
    if ( Re<=Re_l )
        Nu ==Nu_l ;
    elseif ( Re>=Re_t )
        Nu ==Nu_t ;
    else %transition
        Nu ==Nu_l+(Nu_t-Nu_l)*(Re-Re_l)/(Re_t-Re_l) ;
    end
    coh ==Kcoh* lambda*Nu/Dclr; %convective heat transfer coefficient
    Qw ==(Ty - Tmetal) * coh *Ainner; % heat exchange of primary side

   if (Gv>=0)
      Ty.der==(Gv*A.density*Cpin*Tin.T-Gv*B.density*Cpout*Ty-
  Qw)/(denw*Vol*Cpout);
      Tout.T == Ty; % K
    else
      Ty.der==(-Gv*B.density*Cpout*Tout.T+Gv*A.density*Cpin*Ty-
  Qw)/(denw*Vol*Cpin);
      Tin.T == Ty; % K
    end
    Cpin2 ==tablelookup(T_y, Cp_y, Tin2.T, interpolation=linear,
extrapolation=nearest);
    Cpout2==tablelookup(T_y, Cp_y, Tout2.T,interpolation=linear,
extrapolation=nearest );
    denw2 ==(A2.density +B2.density)/2;
    lambda2==tablelookup(T_y, lambda_y, (Tin2.T+Tout2.T)/2,
interpolation=linear, extrapolation=linear);
    Pr2 ==((A2.viscosity_kin+B2.viscosity_kin)/2)/lambda2*denw2*
tablelookup(T_y,Cp_y,(Tin2.T+Tout2.T)/2,interpolation=linear,extrapolat
ion=nearest); % Pr number
    vel2 ==Gv2 /(pi/4 * Dclr^2); % velocity
    Re2 ==vel2/((A2.viscosity_kin+B2.viscosity_kin)/2)*Dclr;
    Nu_l2 ==corr_l(1)* (Re2.^corr_l(2))*(Pr2.^corr_l(3)) ;
    Nu_t2 ==corr_t(1)* (Re2.^corr_t(2))*(Pr2.^corr_t(3)) ;
    if ( Re2<=Re_l )
        Nu2 ==Nu_l2 ;
    elseif ( Re2>=Re_t )
```

```
        Nu2 ==Nu_t2 ;
    else %transition
        Nu2 ==Nu_l2+(Nu_t2-Nu_l2)*(Re2-Re_l)/(Re_t-Re_l) ;
    end
    coh2 ==Kcoh2* lambda2*Nu2/Dclr; % convective heat transfer
coefficient
    Qw2 ==(Tmetal - Ty2) * coh2 *Ainner; % heat exchange of secondary
side
     if (Gv2>=0)
         Ty2.der ==
(Gv2*A2.density*Cpin2*Tin2.T-
Gv2*B2.density*Cpout2*Ty2+Qw2)/(denw2*Vol*Cpout2);
         Tout2.T == Ty2;
    else
         Ty2.der == (-
Gv2*B2.density*Cpout2*Tout2.T+Gv2*A2.density*Cpin2*Ty2+Qw2)/(denw2*Vol*
Cpin2);
         Tin2.T == Ty2;
    end
    Ql == epsi* Sigma0*Aouter*(Tmetal^4 - Tamb^4);
    Tmetal.der == (Qw - Qw2 -Ql)/(mclr*Cpclr);
    dp == p;  % pressure difference
    Tclr ==Tmetal;
  end

end
```

B.3 Code of the Pressurizer block

```
component pctf_pressurizer
% Pressurizer
% This block represents the model of the Pressurizer.
%
% Vol : inner volume of pressurizer, m^3 \n
% Lvl : water level, mm \n
% L3 : water level, 1/100 \n
% pw : pressure of water, Pa \n
% pa : pressure of air, Pa \n
% dena : density of air, kg/m^3 \n
% Ta : air temperature, K \n
% vw : volume of water, m^3 \n
% qw_i : flow rate of water in, m^3/s \n
% qw_o : flow rate of water out, m^3/s \n
% qa_i : flow rate of air in, kg/s \n
% qa_o : flow rate of air out, kg/s \n
% denw : density od water, kg/m^3 \n
% theta : theta of water-convered \n
%
% Port Ain & Aout: air input and output(spill).\n
% Port Win & Wout: water input and output.\n
%
% Copyright 2014-2015 UWO.ECE.CIES

  nodes
    Ain = foundation.pneumatic.pneumatic; % Ain:left
    Aout= foundation.pneumatic.pneumatic; % Aout:left
```

```
    Wout= foundation.hydraulic.hydraulic; % Wout:left
    Win = foundation.hydraulic.hydraulic; % Win:left
  end

  outputs
    Vol = { 0, 'm^3' };  % Vol
    Lvl = { 0.0,  'mm'}; % Lvl
    L3 = {0, '1'};       % L3(%)
    pw ={ 0,  'Pa'};     % pw
    pa ={ 0,  'Pa'};     % pa
    dena ={ 1.293, 'kg/m^3'}; % dena
    Ta ={ 288.15, 'K'};    % Ta
    vw ={0, 'm^3'};        % vw
    qw_i = { 0, 'm^3/s' }; % qw_i
    qw_o = { 0, 'm^3/s' }; % qw_o
    qa_i = { 0, 'kg/s' };  % qa_i
    qa_o = { 0, 'kg/s' };  % qa_o
    denw ={ 1000, 'kg/m^3'}; % denw
    theta ={1, '1'} ;  % theta
  end

  variables
    qa_in  = { 0 , 'kg/s' };  % air into tank
    heat_flow = { 0, 'J/s' }; % for network
    q_spill = { 0, 'kg/s' };  % air that spills out tank
    qw_in = { 0, 'm^3/s' }; % water flowed in
    qw_out = { 0, 'm^3/s' };% water flow out
    Vw = { 0, 'm^3' }; % volume of water
    Va = { 0, 'm^3' }; % volume of air
    denAir = { 1.29228367, 'kg/m^3' }; % density of air
    ma = {0.1, 'kg'} ;% air mass
  end

  parameters
    Dpr = { 310, 'mm' }; % diameter of pressurizer
    Hpr = { 450, 'mm' }; % length of pressurizer
    g ={ 9.80665, 'N/kg'};% gravity acceleration
    Lvl0 = { 10, 'mm' };    % initial level of water
    TAir0 = { 288.15, 'K'}; % initial air temperature in pressurizer
  end
  parameters(Access =private)
    t_unit = { 1 , 's' };  % uniform units
    T_std = { 273.15, 'K'};% standard temperature in Kelvin, = 0C
    m_zero ={ 0, 'kg'} ;    % zero mass
    V_zero ={ 0, 'm^3' };  % zero volume
  end

  function setup
    Vw =0.5*Dpr^2/4 *((2*acos(1-2*Lvl0/Dpr))^3/6)*Hpr;% correspond to 
level computation
    denAir={value(101325/(287.05*TAir0),'1/K'),'kg/m^3'}; % ensure 
initial absolute p=101325Pa
    ma=(pi/4*Dpr^2*Hpr-0.5*Dpr^2/4 *((2*acos(1-
2*Lvl0/Dpr))^3/6)*Hpr)*{value(101325/(287.05*TAir0),'1/K'),'kg/m^3'};
  end
```

```
  branches
    qa_in : Ain.G -> *;
    heat_flow : Ain.Q -> *;
    q_spill : Aout.G -> * ; % negative
    qw_in : Win.q -> *;
    qw_out : Wout.q -> *;    % negative
  end

  equations
    Vol == pi/4*Dpr^2 *Hpr ;
    Vw.der ==qw_in + qw_out;
    Va == Vol - Vw ;
    ma.der == qa_in+q_spill ;
    denAir == ma/Va;

    if (Vw<=pi/2*Dpr^2 *Hpr)
        theta == power( 6* Vw/(0.5* Dpr^2/4 *Hpr),1/3) ;
        Lvl == Dpr/2*(1-cos(theta/2)) ;
    else
        theta == power( 6* Va/(0.5* Dpr^2/4 *Hpr),1/3) ;
        Lvl == Dpr-Dpr/2*(1-cos(theta/2)) ;
    end
    L3 == Lvl/Dpr*100; % water level with unit %
    Ain.T == Ain.Ta;
    Ta == Ain.T;
    pa == denAir* Ta *Ain.R -Aout.Pa; % gage pressure
    pw == Win.density * Lvl *g ; % water pressure at the bottom

    Ain.p == pa + Ain.Pa ; % absolute p
    Aout.p== pa + Aout.Pa; % absolute p
    Win.p == pa + pw; % gage p
    Wout.p== pa + pw; %
    dena == denAir ;  % for output
    denw == Win.density ;% monitor
    vw == Vw ;          % output
    qw_i == qw_in ;     % monitor
    qw_o == -qw_out;    % direction is outward, m^3/s
    qa_i == qa_in;      % air flow rate of ingoing, kg/s
    qa_o == -q_spill;   % air flow rate of outgoing, kg/s

  end

end
```

B.4 Code of the HX-Tank block

```
component pctf_hxtank
% HX-Tank
% This block represents the model of the HX-Tank.
%
% Vol : inner volume of tank, m^3' \n
% Lvl : water level, mm \n
% L4 : water level, 1/100 \n
% pw : pressure of water, Pa \n
% pa : pressure of air, Pa \n
% dena : density of air, kg/m^3 \n
```

```
% Ta : air temperature, K \n
% vw : volume of water, m^3 \n
% qw_i : flow rate of water into tank, m^3/s \n
% qw_o : flow rate of water out of tank, m^3/s \n
% qa_i : flow rate of air into tank, kg/s \n
% qa_o : flow rate of air out of tank, kg/s \n
% denw : density of water, kg/m^3 \n
%
% Port Ain & Aout: air input and output (spill).\n
% Port Win & Wout: water input and output.\n
%
% Copyright 2014-2015 UWO.ECE.CIES

  nodes
    Aout= foundation.pneumatic.pneumatic; % Aout:left
    Win = foundation.hydraulic.hydraulic; % Win:left
    Wout= foundation.hydraulic.hydraulic; % Wout:left
    Ain = foundation.pneumatic.pneumatic; % Ain:left
  end

  outputs
    Vol = { 0, 'm^3' };  % Vol
    Lvl = { 0.0,  'mm'}; % Lvl
    L4 = { 0.0,  '1'}; % L4(%)
    pw = { 0,  'Pa'};   % pw
    pa = { 0,  'Pa'};   % pa
    dena = { 1.293, 'kg/m^3'}; % dena
    Ta = { 288.15, 'K'}; %Ta
    vw = {0, 'm^3'};         %vw
    qw_i = { 0, 'm^3/s' }; %qw_i
    qw_o = { 0, 'm^3/s' }; %qw_o
    qa_i = { 0, 'kg/s' };  %qa_i
    qa_o = { 0, 'kg/s' };  %qa_o
    denw = { 1000, 'kg/m^3'}; % denw
  end

  variables
    qa_in  = { 0 , 'kg/s' };  % air into tank
    heat_flow = { 0, 'J/s' }; % for network
    q_spill = { 0, 'kg/s' };  % air that spills out tank
    qw_in = { 0, 'm^3/s' }; % water flowed in
    qw_out = { 0, 'm^3/s' };% water flow out
    crossA = { 1, 'm^2' }; % cross-sectional area of tank
    Vw = { 0, 'm^3' }; % volume of water
    Va = { 0, 'm^3' }; % volume of air
    denAir = { 1.29228367, 'kg/m^3' }  ; % density of air at 0C
    Vbubble ={ 0, 'm^3' }; % volume of bubble
    ma = {0.1, 'kg'} ;% air mass
  end

  parameters
    Dhx = { 310, 'mm' }; % diameter of tank
    Hhx = { 450, 'mm' }; % height of tank
    schi ={ 0.1, '1'};   % bubble coefficient
    g ={ 9.80665, 'N/kg'};% gravity acceleration
    Lvl0 = { 10, 'mm' };  % initial level of water
```

```
    TAir0 = { 288.15, 'K'}; %initial air temperature in tank
  end
  parameters(Access =private)
    t_unit = { 1 , 's' };  % uniform units
    T_std = { 273.15, 'K'};% standard temperature in Kelvin, = 0C
    coBubble = { 5, 'kg/m^3'}; % used as the coeff to balance pressure
effect
  end

  function setup
    % Parameter range checking
    if Lvl0 > Hhx
      pm_error('simscape:GreaterThanTankHeight','initial height of
water')
      Lvl0 =Hhx ;
    end
    Vw = pi/4*Dhx^2*Lvl0 ;% initial volume of water as initial
condition of Vw differential
    denAir = {value(101325/(287.05*TAir0),'1/K'),'kg/m^3'}; % ensure
initial absolute p=101325Pa
    ma = (pi/4*Dhx^2*Hhx-
pi/4*Dhx^2*Lvl0)*{value(101325/(287.05*TAir0),'1/K'),'kg/m^3'};
  end

  branches
    qa_in : Ain.G -> *;
    heat_flow : Ain.Q -> *;
    q_spill : Aout.G -> * ; % negative
    qw_in : Win.q -> *;
    qw_out : Wout.q -> *;   % negative
  end

  equations
    crossA == pi/4*Dhx^2 ;
    Vol == pi/4*Dhx^2*Hhx ;
    Vw.der == qw_in + qw_out;
    Va == Vol - Vw ;
    ma.der == qa_in+q_spill ;
    denAir == ma/Va;

    Ain.T== Ain.Ta;
    if qa_in> 0
        Ta == Ain.T;
    else
        Ta == Ain.Ta;
    end
    pa ==denAir* Ta *Ain.R -Aout.Pa;  %gage
    Vbubble ==schi*abs(qa_in)/(denAir^4)*coBubble^3 *t_unit ;
    if (Vbubble>Vw)
        Lvl == (Vw+Vw)/crossA ;
    else
        Lvl == (Vw+Vbubble)/crossA ;
    end
    L4 ==Lvl/Hhx*100; % water level with unit %
```

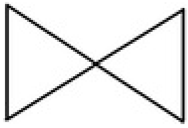
```
    pw == Win.density * Vw/crossA *g ; % water pressure at the bottom
of tank
    Ain.p == pa + pw + Ain.Pa ;% absolute p, where air enters at bottom
of the tank
    Aout.p == pa + Aout.Pa ; % absolute p
    Win.p == pa + pw; % gage p
    Wout.p== pa + pw; %

    dena == denAir ;
    denw == Win.density ;
    vw == Vw ;            % for output
    qw_i == qw_in ;
    qw_o == -qw_out;      % direction is outward, m^3/s
    qa_i == qa_in;        % air flow rate of ingoing, kg/s
    qa_o == -q_spill;     % air flow rate of outgoing, kg/s
  end

end
```

B.5 Code of the Turbine block

```
component pctf_turbine
% Turbine & Generator
% This block represents the simple model of the Turbine.
%
% Fin_fortest : just for test mode, air flow rate = the input, kg/s \n
% mode : test mode switch, 0: normal; 1: test mode, \n
% speed : speed of turbine, rad/s \n
% power : power of generator, W \n
% qa_i : flow rate of air through turbine, kg/s \n
%
% Copyright 2014-2015 UWO.ECE.CIES

  nodes
    A = foundation.pneumatic.pneumatic; % A:left
    B = foundation.pneumatic.pneumatic; % B:left
  end

  inputs
    Fin = { 0.108 , 'kg/s' };   % Fin_fortest
    mode = { 0, '1'};           % mode
  end

  outputs
    speed = { 0, 'rad/s' };   % speed
    power = { 0, 'W'};        % power
    qa_i = { 0, 'kg/s' };   %qa_i
  end

  variables
    qa = { 0, 'kg/s' };    % flow rate of air
    pa = { 0,  'Pa'};      % pressure of air
    resistance = { 0.5e+7, 'Pa/(kg/s)' }; % real resistance
    heat_flow = { 0, 'J/s' };             % for network
    temperature_difference = { 0 , 'K' }; % for network
```

```
    qin = { 0, 'kg/s' };  % input air flow rate
    rpm = { 0, 'rad/s' }; % internal quantity
  end

  parameters
    resisF = { 0.5e+7, 'Pa/(kg/s)' }; % maximum resistance
    Jw = { 5.e-5, 'kg*m^2' }; % moment of inertia
    f = { 1.7e-5, 'kg*m^2' }; % viscous friction
    ita = { 0.72, '1' }; % conversion rate from kinetic momentum, for
adjusting speed
    k = { 0.95, '1' };   % coefficient of energy conversion to
electricity, for adjusting output power
    speed0 = { 0, 'rad/s' };  % initial speed
  end
  parameters(Access =private)
    t_unit = { 1 , 's' };  % uniform unit
    r_unit = { 1 , 'm' };  % uniform unit
    P_zero ={ 0, 'W' };          % zero power
    rpm_max = { 1300, 'rad/s' }; % maximum speed
  end

  function setup
    rpm = speed0;
  end

  branches
    qa : A.G -> B.G;
    heat_flow : A.Q -> B.Q;
  end

  equations
    temperature_difference == A.T - B.T;
    pa == A.p - B.p;

    qin == if (mode==0),qa else Fin end; % mass flow rate, kg/s
    t_unit*Jw*rpm.der == ita*qin*r_unit^2 - f*rpm;

    if (qa>0)
        power =={ value(k*f*rpm*1000,'kg*m^2*rad/s'),'W' } ;
    else
        power ==P_zero ;
    end
    speed ==rpm;
    resistance == resisF*abs(1-(rpm/rpm_max)) ;
    pa == resistance * qa;
    heat_flow == qa*(A.c_p+B.c_p)/2*temperature_difference ;
    qa_i ==qin ;
  end

end
```

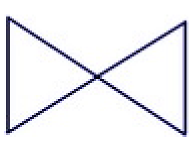
B.6 Code of the Water Valve block

Image: 

```
component pctf_valve
% Valve
% This block represents the model of the custom water valve. It is
% simplified to suit the simulation system. The valve characteristic
% curve may be specified.
%
% Pos : the input of open position signal, \n
% F : water flow rate, kg/s \n
% dp : pressure difference between inlet and outlet, Pa \n
%
% Valve characteristic curve can be specified by vectors.
% Connections A and B are conserving hydraulic ports associated
% with the block inlet and outlet, respectively. The block positive
% direction is from port A to port B. This means that the flow rate is
% positive if fluid flows from A to B,and the pressure loss is
% determined as dp = p_A - p_B.
%
% Copyright 2014-2015 UWO.ECE.CIES

  nodes
    A = foundation.hydraulic.hydraulic; % A:left
    B = foundation.hydraulic.hydraulic; % B:right
  end

  inputs
    Pos = { 1, '1'};
  end
  outputs
    Fm  ={ 1,   'kg/s'};    % F
    dp ={ 0,   'Pa'};       % dp
  end

  variables
    q = { 1, 'm^3/s' };
    p = { 0 , 'Pa' };
    resistance = { 1.e+7, 'Pa/(m^3/s)' }; % hydraulic resistance
    areaO = { 1, '1'};  % crossed area
    density = { 1.e+3, 'kg/m^3' };% density of water
  end

  parameters
    resisF = { 1.e+7, 'Pa/(m^3/s)' }; % hydraulic resistance of full-
open
    CoeSen =  { 0.5, '1'}; %coefficient of area to resistance (between
0-1)
    minFlowRate = { 0, 'm^3/s' }; % the flow rate when fully closed
    ifFullClose =  { 0, '1'};  % full close selection, 0: normal; 1:
full close
  end
```

```
  parameters (Size=variable)  % Lookup Table as valve curve
   xd = {[0 0.1 0.2 0.3 0.4 0.5 0.6 0.7 0.8 0.9 1] '1'};% valve curve
x-axis
   yd = {[0 0.1 0.2 0.3 0.4 0.5 0.6 0.7 0.8 0.9 1] '1'};% valve curve
y-axis
 end

  function setup
    % Parameter range checking
    if CoeSen < 0 || CoeSen >1
      error('sensitivity should be greater than 0 and less than 1');
      CoeSen=0.5;
    end
  end

  branches
    q : A.q -> B.q ;
  end

  equations
     p == A.p - B.p;
     density ==A.density; % from loop assigned
     if Pos <=0
         areaO == 0;
         resistance == resisF/(1-CoeSen*(1-areaO));
     elseif Pos >1.0
         areaO == 1;
         resistance == resisF/(1-CoeSen*(1-areaO));
     else
         areaO==tablelookup(xd, yd, Pos, interpolation=linear,
extrapolation=nearest);
         resistance == resisF /(1-CoeSen*(1-areaO));
     end
     if (ifFullClose>0.5)
         q == minFlowRate ;
     else
         q == p/resistance ;
     end
     Fm  == q * density;  % kg/s
     dp == p;
  end

end
```

B.7 Code of the Air Valve block

Image: 

```
component pctf_airvalve
% Air Valve
% This block represents the model of the custom air valve. It is
simplified
% to suit the simulation system. The valve characteristic curve may be
```

```
% specified.
%
% Pos : the input of open position signal in uniform.\n
% dp : pressure difference between inlet and outlet, Pa \n
%
% Valve characteristic curve can be specified by vectors.
% dp = p_A - p_B.
%
% Copyright 2014-2015 UWO.ECE.CIES

  nodes
    A = foundation.pneumatic.pneumatic; % A:left
    B = foundation.pneumatic.pneumatic; % B:right
  end

  inputs
    Pos = { 1, '1'};
  end

  outputs
    dp ={ 0,  'Pa'};      % dp
  end

  variables
    q = { 1, 'kg/s'  };
    p = { 0 , 'Pa' };
    resistance = { 1.e+7, 'Pa/(kg/s)' }; % real resistance
    areaO = { 1, '1'};  % crossed area
    heat_flow = { 0, 'J/s' };              % for network
    temperature_difference = { 0 , 'K' }; % for network
  end

  parameters
    resisF = { 1.e+7, 'Pa/(kg/s)' }; % resistance of full-open
    CoeSen =  { 0.5, '1'}; % coefficient of area to resistance (between
0-1)
    deadzone_lower = { 0.2, '1' }; % dead zone lower margin
    deadzone_upper = { 0.8, '1'};  % dead zone upper margin
  end
  parameters (Size=variable)  % Lookup Table as valve curve
   xd = {[0 0.1 0.2 0.3 0.4 0.5 0.6 0.7 0.8 0.9 1] '1'};% valve curve
x-axis
   yd = {[0 0.1 0.2 0.3 0.4 0.5 0.6 0.7 0.8 0.9 1] '1'};% valve curve
y-axis
  end

  function setup
    % Parameter range checking
    if CoeSen < 0 || CoeSen >1
      error('sensitivity should be greater than 0 and less than 1');
      CoeSen=0.5;
    end
  end

  branches
```

```
    q : A.G -> B.G ;
    heat_flow : A.Q -> B.Q;
  end

  equations
     temperature_difference == A.T - B.T;
     p == A.p - B.p;
     if Pos <= deadzone_lower
         areaO == 0;
         resistance == resisF/(1-CoeSen*(1-areaO));
     elseif Pos >= deadzone_upper
         areaO == 1;
         resistance == resisF/(1-CoeSen*(1-areaO));
     else
         areaO==tablelookup(xd, yd, (Pos-deadzone_lower),
interpolation=linear, extrapolation=nearest);
         resistance == resisF /(1-CoeSen*(1-areaO));
     end
     p == resistance * q;
     heat_flow == q*(A.c_p+B.c_p)/2*temperature_difference ;
     dp == p;
  end

end
```

B.8 Code of the Thermal Inertia block

```
component pctf_thermalinertia
% Thermal inertia
% This block is used to simulate thermal inertia in pipes.
%
% Gv : water flow rate, m^3/s \n
% Tamb : the input of ambient temperature, K \n
% Ql : heat lost to ambient, J/s \n
% dT : temperature difference between inlet and outlet, K \n
% td : time delay, s \n
%
% Nodes Tin, Tout : thermal conserving ports.
%
% Copyright 2014-2015 UWO.ECE.CIES

 nodes
    Tin = foundation.thermal.thermal; % Tin:right
    Tout= foundation.thermal.thermal; % Tout:left
 end

 inputs
    Gv = { 1, 'm^3/s' };       % Gv
    Tamb = { 293.15, 'K' };    % Tamb:left
 end

  outputs
    Ql ={ 0,  'J/s'}; % Ql:right
    dT ={ 1 , 'K' };  % dT:right
    td ={ 1 , 's' };  % td:right
  end
```

```
  variables
    Ty = { 293.15 , 'K' };  % outlet temperature
    Ttube = { 293.15 , 'K' };  % tube temperature
    Qs = { 0, 'J/s' }; %recerved
    Ts = { 0, 'K' };   %reserved
    denw = { 1.e+3, 'kg/m^3' };%density of water
    Cpin = { 4.18e+3, 'J/(kg*K)' };    % specific heat of inlet water
    Cpout= { 4.18e+3, 'J/(kg*K)' };    % specific heat of outlet water
    Vol = { 0, 'm^3' };          % inter volume
    area = { 0, 'm^2'};          % surface area
  end

  parameters
    Diameter = { 20, 'mm' };  % diameter
    Length = { 100, 'mm' };    % length
    coh ={0, 'W/(m^2*K)'};     % heat transfer coefficient of heat lost
    epsi = { 0, '1' };         % coefficient of heat lost to ambient
    T0 = { 293.15 , 'K' };     % initial temperature
  end
  parameters(Access =private)
    t_unit = { 1 , 's' };  % uniform units
    Sigma0 ={ 5.67e-8, 'W/(m^2*K^4)' }; % Stefan-Boltzmann constant
    G_unit = { 0.01 , 'm^3/s'}; %
    T_y={[273.16:10:373.16],'K'}; % 11
    Cp_y={[4.212 4.191 4.183 4.174 4.174 4.174 4.178 4.178 4.195 4.208
4.220],'J/(g*K)'};
    denw_y={[999.9 999.7 998.2 995.7 992.2 988.1 983.2 977.8 971.8
965.3 958.4], 'kg/m^3'};
  end

  function setup
    Ttube = T0; % ensure correct initial conditions
    Ty = T0;
  end

  branches
    Qs : Tin.Q -> Tout.Q;
  end

  equations
     Ts == Tin.T - Tout.T;
     Vol == pi/4*Diameter^2*Length;
     area== pi*Diameter*Length;
     Ttube == (Tin.T+Tout.T)/2;
     if (Ttube > Tamb)
        Ql == epsi* Sigma0*area*(Ttube^4 - Tamb^4)+coh *area*(Ttube-
Tamb);
     else
        Ql == epsi* Sigma0*area*(Ttube^4 - Tamb^4);
     end
     Cpin ==tablelookup(T_y, Cp_y, Tin.T, interpolation=linear,
extrapolation=nearest);
     Cpout==tablelookup(T_y, Cp_y, Tout.T,interpolation=linear,
extrapolation=nearest );
```

```
     denw ==tablelookup(T_y, denw_y, Ttube,interpolation=linear,
extrapolation=linear );
     if (Gv>=0)
        Vol*denw*Cpout*Ty.der ==
    tablelookup(T_y,denw_y,Tin.T)*Gv*Cpin*Tin.T-
    tablelookup(T_y,denw_y,Tout.T)*Gv*Cpout*Ty-Ql;
        Tout.T == Ty;
     else
        Vol*denw*Cpin*Ty.der==-
    tablelookup(T_y,denw_y,Tout.T)*Gv*Cpout*Tout.T+tablelookup(T_y,denw_
    y,Tin.T)*Gv*Cpin*Ty-Ql;
        Tin.T == Ty;
     end
     td ==Vol/(abs(Gv)+G_unit);
     dT ==Ts;
  end

end
```

Appendix C: Port Definitions of Custom-defined Blocks

C.1 Port definition of the Heater block

Type	Direction	Denotation	Description	Default Unit
Conserving ports	N/A	*A, B*	Conserving hydraulic ports, associated with the block inlet and outlet, respectively.	
	In/Out	*Tin, Tout*	Conserving thermal ports, connecting temperatures at inlet and outlet.	
Signal ports	Input	*I*	connecting to current signal	*A*
		Tamb	input of ambient temperature	*K*
		Gin_fortest	only for test mode, in which the flow rate = the input	*m^3/s*
		mode	test mode switch, 0: normal; 1: test mode	
	Output	*Vol*	inner volume of heater	*m^3*
		Gv	water flow rate	*m^3/s*
		Tshh	temperature of metal sheath	*K*
		Qw	heat transfer flux between water and sheath	*J/s*
		Ql	heat lost to ambience	*J/s*
		dp	pressure difference between inlet and outlet	*Pa*
		denw	density of water	*kg/m^3*
		Pr	Prandtl number	
		vel	velocity of water	*m/s*
		Re	Reynolds number	

		Nu	Nusselt number	
		coh	convection heat transfer coefficient	*W/(m^2*K)*

C.2 Port definition of the Chiller block

Type	Direction	Denotation	Description	Default Unit
Conserving ports	N/A	*A, B*	Conserving hydraulic ports, connecting to primary side as inlet and outlet.	
	N/A	*A2, B2*	Conserving hydraulic ports, connecting to secondary side as inlet and outlet.	
	In/Out	*Tin, Tout*	Conserving thermal ports, connecting temperatures at inlet and outlet of primary side.	
	In/Out	*Tin2, Tout2*	Conserving thermal ports, connecting temperatures at inlet and outlet of secondary side.	
Signal ports	Input	*Tamb*	the input of ambient temperature	*K*
		Gin_fortest	only for test mode, in which the flow rate of primary side = the input	*m^3/s*
		Gin2_fortest	just for test mode, in which the flow rate of secondary side = the input	*m^3/s*
	Output	*Gv*	water flow rate of primary side	*m^3/s*
		Gv2	water flow rate of secondary side	*m^3/s*
		Tclr	the temperature of metal lamellas	*K*
		Qw	heat transfer flux of primary side	*J/s*
		Ql	heat lost to ambient	*J/s*
		dp	pressure difference between inlet	*Pa*

			and outlet of primary side	
		denw	density of water	*kg/m^3*
		Pr	Prandtl number of primary side	
		vel	velocity of water of primary side	*m/s*
		Re	Reynolds number of primary side	
		Nu	Nusselt number of primary side	
		coh	convective heat transfer coefficient of primary side	*W/(m^2*K)*
		Pr2	Prandtl number of secondary side	
		vel2	velocity of water of secondary side	*m/s*
		Re2	Reynolds number of secondary side	
		Nu2	Nusselt number of secondary side	
		coh2	convective heat transfer coefficient of secondary side	*W/(m^2*K)*

C.3 Port definition of the Pressurizer block

Type	Direction	Denotation	Description	Default Unit
Conserving ports	In/Out	*Ain, Aout*	Conserving pneumatic ports as air input and output (spill).	
	In/Out	*Win, Wout*	Conserving hydraulic ports as water input and output.	
Signal ports	Output	*Vol*	inner volume of pressurizer	*m^3*
		Lvl	water level	*mm*
		L3	water level	*%*
		pw	gage pressure of water	*Pa*
		pa	gage pressure of air	*Pa*
		dena	density of air	*kg/m^3*

		Ta	air temperature	*K*
		vw	volume of water	*m^3*
		qw_i	flow rate of water in	*m^3/s*
		qw_o	flow rate of water out	*m^3/s*
		qa_i	flow rate of air in	*kg/s*
		qa_o	flow rate of air out	*kg/s*
		denw	density of water	*kg/m^3*
		theta	angle of water-covered	

C.4 Port definition of the HX-Tank block

Type	Direction	Denotation	Description	Default Unit
Conserving ports	In/Out	*Ain, Aout*	Conserving pneumatic ports as air input and output (spill).	
	In/Out	*Win, Wout*	Conserving hydraulic ports as water input and output.	
Signal ports	Output	*Vol*	inner volume of pressurizer	*m^3*
		Lvl	water level	*mm*
		L4	water level	*%*
		pw	pressure of water	*Pa*
		pa	pressure of air	*Pa*
		dena	density of air	*kg/m^3*
		Ta	air temperature	*K*
		vw	volume of water	*m^3*
		qw_i	flow rate of water into tank	*m^3/s*
		qw_o	flow rate of water out of tank	*m^3/s*
		qa_i	flow rate of air into tank	*kg/s*

		qa_o	flow rate of air out of tank	*kg/s*
		denw	density of water	*kg/m^3*

C.5 Port definition of the Turbine block

Type	Direction	Denotation	Description	Default Unit
Conserving ports	N/A	*A, B*	Conserving pneumatic ports as air inlet and outlet.	
Signal ports	Input	*Fin_fortest*	only for test mode, flow rate =the input	*kg/s*
		mode	test mode switch, 0: normal; 1: test mode	
	Output	*speed*	speed of turbine	*rpm*
		power	power of generator	*W*
		qa_i	mass flow rate of air through turbine	*kg/s*

C.6 Port definition of the Water Valve and Air Valve block

1. Water Valve

Type	Direction	Denotation	Description	Default Unit
Conserving ports	N/A	*A, B*	Conserving hydraulic ports, associated with the block inlet and outlet, respectively.	
Signal ports	Input	*Pos*	input of position signal	
	Output	*F*	mass flow rate of water	*kg/s*
		dp	pressure difference between inlet and outlet	*Pa*

2. Air Valve

Type	Direction	Denotation	Description	Default Unit

Conserving ports	N/A	*A, B*	Conserving hydraulic ports, associated with the block inlet and outlet, respectively.	
Signal ports	Input	*Pos*	input of position signal	
		dp	pressure difference between inlet and outlet	*Pa*

C.7 Port definition of the Thermal Inertia block

Type	Direction	Denotation	Description	Default Unit
Conserving ports	In/Out	*Tin, Tout*	Conserving thermal ports, connecting temperatures at inlet and outlet.	
Signal ports	Input	*Gv*	volumetric flow rate of water	*m^3/s*
		Tamb	input of ambient temperature	*K*
	Output	*Ql*	heat lost to ambience	*J/s*
		dT	temperature difference between inlet and outlet	*K*
		td	time delay	*s*

Appendix D: Parameter Setting used in Main Components

D.1 Parameters of the Heater

parameter	description	unit	value
resisdefault	hydraulic resistance	*Pa/(m^3/s)*	7.5e+7
ita	efficiency of energy conversion		0.97
Ue	voltage of electrical source	*V*	230
Dhtr_o	outer diameter of heater	*in*	2.5
Dhtr	inner diameter of heater	*in*	1.5
Dshh	diameter of sheath	*in*	0.315
Hhtr	height of heater	*in*	20
Him	height immersed	*in*	18
mshh	equivalent mass of sheath	*kg*	4.0
Cpshh	specific heat of sheath	*J/kg/K*	460
epsi	coefficient of heat lost to ambient		0.99
corr_l	Nusselt number correlation coefficients of laminar regime		[0.664 0.5 0.33]
corr_t	Nusselt number correlation coefficients of turbulent regime		[1.56 0.85 0.33]
Re_l	laminar flow upper margin		1000
Re_t	turbulent flow lower margin		2300
Tao_e	time constant of energy conversion	*s*	10

D.2 Parameters of the Chiller

parameter	description	unit	value
resisdefault	hydraulic resistance of primary side	*Pa/(m^3/s)*	9.5e+7
resisdefault2	hydraulic resistance of secondary side	*Pa/(m^3/s)*	9.e+7
Vol	inner volume	*l*	0.5
Ainner	inner surface area	*mm^2*	3.e+5
Aouter	outer surface area	*mm^2*	50000
Dclr	hydraulic diameter	*mm*	35
Lclr	length of flow path	*mm*	200
mclr	mass of lamella	*kg*	5.0
Cpclr	specific heat of lamella	*J/kg/K*	500
epsi	coefficient of heat lost to ambient		0.99
corr_l	Nusselt number correlation coefficients of laminar regime		[0.664 0.8 0.33]
corr_t	Nusselt number correlation coefficients of turbulent regime		[0.316 0.8 0.33]
Re_l	laminar flow upper margin		2000
Re_t	turbulent flow lower margin		4000

D.3 Parameters of the Pressurizer

parameter	description	unit	value
Dpr	diameter of pressurizer	*mm*	310
Hpr	length of pressurizer	*mm*	450
g	gravity acceleration	*N/kg*	9.80665

D.4 Parameters of the HX-Tank

parameter	description	unit	value
Dhx	diameter of tank	*mm*	310
Hhx	height of tank	*mm*	450
schi	bubble coefficient		0.28
g	gravity acceleration	*N/kg*	9.80665

D.5 Parameters of the Turbine

parameter	description	unit	value
resisF	maximum resistance	*Pa*s/kg*	2.5e+7
Jw	moment of inertia	*kg*m^2*	4.5e-5
f	viscous friction	*kg*m^2*	1.7e-5
ita	conversion rate from kinetic momentum, for adjusting speed		0.7
k	coefficient of energy conversion to electricity, for adjusting output power		0.95

Appendix E: Scheduled Validation Cases

Case 1.1 Validation of the Heater model

Purpose of validation	The response from the current (input power) increasing to the output temperature.
Time	150 s
Condition of simulation and experiment	Tamb (ambient temperature) : 24 ℃
Disturbance	C2 (current): 0->30 (A)
Operation in experiment	Maintain F1 (water flow rate) stable at 6 l/Min Increase C2 (current) from 0 to 30 A.
Simulation mode	Test mode
Simulation input	T1 (inlet temperature), by using Table Block.
	F1 (flow rate), by using Table Block.
	C2 (current), by using Table Block.
Output for comparison	T2 (outlet temperature)
Operation in simulation	Identify the Nusselt number correlation coefficients, time constant.
Evaluative method	T2: quantitative assessment with absolute error, relative error.
Illustration	Condition: T1, F1; C2
	Result: T2

Case 1.2 Validation of the Heater model

Purpose of validation	The response from the current (input power) decreasing to the output temperature.
Time	150 s
Condition of simulation and experiment	Tamb (ambient temperature) : 24 ℃

Disturbance	C2 (current): 30->0 (A)
Operation in experiment	Maintain F1 (water flow rate) stable at 6 l/Min Decrease C2 (current) from 30 to 0 (A).
Simulation mode	Test mode
Simulation input	T1 (inlet temperature), by using Table Block.
	F1 (flow rate), by using Table Block.
	C2 (current), by using Table Block.
Output for comparison	T2 (outlet temperature)
Operation in simulation	Identify the Nusselt number correlation coefficients, time constant.
Evaluative method	T2: quantitative assessment with absolute error, relative error.
Illustration	Condition: T1, F1; C2
	Result: T2

Case 1.3 Validation of the Heater model

Purpose of validation	The response from the flow rate increasing to the output temperature.
Time	150 s
Condition of simulation and experiment	Tamb (ambient temperature) : 24 ℃
Disturbance	F1 (water flow rate): 4->9 (l/Min)
Operation in experiment	Maintain C2 (current) at 30 A. Increase F1 (water flow rate) from 4 to 9 (l/Min), by opening CV-1 (water inlet valve).
Simulation mode	Test mode
Simulation input	T1 (inlet temperature), by using Table Block.

	F1 (flow rate), by using Table Block.
	C2 (current), by using Table Block.
Output for comparison	T2 (outlet temperature)
Operation in simulation	Identify the Nusselt number correlation coefficients, time constant.
Evaluative method	T2: quantitative assessment with absolute error, relative error.
Illustration	Condition: T1, C2; F1
	Result: T2

Case 1.4 Validation of the Heater model

Purpose of validation	The response from the flow rate decreasing to the output temperature.
Time	150 s
Condition of simulation and experiment	Tamb (ambient temperature) : 24 ℃
Disturbance	F1 (water flow rate): 9->4 (l/Min)
Operation in experiment	Maintain C2 (current) at 30 A. Decrease F1 (water flow rate) from 9 to 4 (l/Min), by closing CV-1 (water inlet valve).
Simulation mode	Test mode
Simulation input	T1 (inlet temperature), by using Table Block.
	F1 (flow rate), by using Table Block.
	C2 (current), by using Table Block.
Output for comparison	T2 (outlet temperature)
Operation in simulation	Identify the Nusselt number correlation coefficients, time constant.
Evaluative method	T2: quantitative assessment with absolute error, relative error.

Illustration	Condition: T1, C2; F1
	Result: T2

Case 1.5 Validation of the Heater model

Purpose of validation	The response from the inlet temperature increasing to the output temperature.
Time	240 s
Condition of simulation and experiment	Tamb (ambient temperature) : 25 ℃
Disturbance	T1 (inlet temperature): 12->20 (℃)
Operation in experiment	Maintain C2 (current) at 20 A. Increase T1 (inlet temperature) from 10 ℃ to 20 ℃, by closing CV-34 (3-way water valve).
Simulation mode	Test mode
Simulation input	T1 (inlet temperature), by using Table Block.
	F1 (flow rate), by using Table Block.
	C2 (current), by using Table Block.
Output for comparison	T2 (outlet temperature)
Operation in simulation	Identify the Nusselt number correlation coefficients, time constant.
Evaluative method	T2: quantitative assessment with absolute error, relative error.
Illustration	Condition: F1, C2; T1
	Result: T2

Case 1.6 Validation of the Heater model

Purpose of validation	The response from the inlet temperature decreasing to the output temperature.

Time	240 s
Condition of simulation and experiment	Tamb (ambient temperature) : 20 ℃
Disturbance	T1 (inlet temperature): 22->13 (℃)
Operation in experiment	Maintain C2 (current) at 20 A. Decrease T1 (inlet temperature) from 20 ℃ to 10 ℃, by opening CV-34 (3-way water valve).
Simulation mode	Test mode
Simulation input	T1 (inlet temperature), by using Table Block.
	F1 (flow rate), by using Table Block.
	C2 (current), by using Table Block.
Output for comparison	T2 (outlet temperature)
Operation in simulation	Identify the Nusselt number correlation coefficients, time constant.
Evaluative method	T2: quantitative assessment with absolute error, relative error.
Illustration	Condition: F1, C2; T1
	Result: T2

Case 1.7 Validation of the Heater model

Purpose of validation	The response from both the current and the water flow rate to the outlet temperature. Once the input power rising, increasing the fluid flow rate to maintain the temperature of primary coolant.
Time	240 s
Condition of simulation and experiment	Tamb (ambient temperature) : 24 ℃
Disturbance	C2 (current) : 15->30 (A)
Operation in experiment	Increase C2 (current) from 15 to 30 (A) manually in 10% step.

	Increase F1 (water flow rate) from 6 to 9 (l/Min), by opening CV-1 (water inlet valve) to maintain T2 (outlet temperature) at 30 ℃.
Simulation mode	Test mode
Simulation input	T1 (inlet temperature), by using Table Block.
	F1 (flow rate), by using Table Block.
	C2 (current), by using Table Block.
Output for comparison	T2 (outlet temperature)
Operation in simulation	Identify the Nusselt number correlation coefficients, time constant.
Evaluative method	T2: quantitative assessment with absolute error, relative error.
Illustration	Condition: T1, F1; C2
	Result: T2

Case 1.8 Validation of the Heater model

Purpose of validation	The response from both the current and the water flow rate to the outlet temperature. Once the input power is dropping, decreasing the fluid flow rate to maintain the temperature of primary coolant.
Time	240 s
Condition of simulation and experiment	Tamb (ambient temperature) : 24 ℃
Disturbance	C2 (current) : 30->15 (A)
Operation in experiment	Decrease C2 (current) from 30 to 15 (A) manually in 10% step. Decrease F1 (water flow rate) from 9 to 6 (l/Min), by closing CV-1 (water inlet valve) to maintain T2 (outlet temperature) at 30 ℃.
Simulation mode	Test mode
Simulation input	T1 (inlet temperature), by using Table Block.
	F1 (flow rate), by using Table Block.

	C2 (current), by using Table Block.
Output for comparison	T2 (outlet temperature)
Operation in simulation	Identify the Nusselt number correlation coefficients, time constant.
Evaluative method	T2: quantitative assessment with absolute error, relative error.
Illustration	Condition: T1, F1; C2
	Result: T2

Case 2.1 Validation of the Chiller model

Purpose of validation	The response from the inlet temperature of 1st side to output temperature. This temperature rising is caused by the input power. This is to check the heat exchange and heat diffusion of Chiller.
Time	280 s
Condition of simulation and experiment	Tamb (ambient temperature) : 24 ℃
Disturbance	T5 (inlet temperature of 1st side) increasing
Operation in experiment	Maintain F1 (flow rate of 1st side) stable at 6 l/Min by operating CV-1 (water inlet valve). Maintain F3 (flow rate of 2nd side) stable at 10 l/Min by operating the 3-way valve CV-34 in 2nd side. Chiller pump is on. Increase T5 (inlet temperature of 1st side) from 20 ℃ to 35 ℃, by increasing the current C2 of the heater.
Simulation mode	Test mode
Simulation input	F1 (flow rate of 1st side), by using Table Block.
	T5 (inlet temperature of 1st side), by using Table Block.
	F3 (flow rate of 2nd side), preprocessed into smooth data and input by using Table Block.

	T6 (inlet temperature of 2nd side), by using Table Block.
Output for comparison	T3 (outlet temperature of 1st side)
Operation in simulation	Identify the Nusselt number correlation coefficients.
Evaluative method	T3: quantitative assessment with absolute error, relative error.
Illustration	Condition: F1; T5
	Result: T3

Case 2.2 Validation of the Chiller model

Purpose of validation	The response from the inlet temperature of 1st side to output temperature. This temperature dropping is caused by the input power. This is to check the heat exchange and heat diffusion of Chiller.
Time	280 s
Condition of simulation and experiment	Tamb (ambient temperature) : 24 ℃
Disturbance	T5 (inlet temperature of 1st side) decreasing
Operation in experiment	Maintain F1 (flow rate of 1st side) stable at 6 l/Min by operating CV-1 (water inlet valve). Maintain F3 (flow rate of 2nd side) stable at 10 l/Min by operating the 3-way valve CV-34 in 2nd side. Chiller pump is on. Decrease T5 (inlet temperature of 1st side) from 35 ℃ to 20 ℃, by decreasing the current C2 of the heater.
Simulation mode	Test mode
Simulation input	F1 (flow rate of 1st side), by using Table Block.
	T5 (inlet temperature of 1st side), by using Table Block.
	F3 (flow rate of 2nd side), preprocessed into smooth data and input by using Table Block.

	T6 (inlet temperature of 2nd side), by using Table Block.
Output for comparison	T3 (outlet temperature of 1st side)
Operation in simulation	Identify the Nusselt number correlation coefficients.
Evaluative method	T3: quantitative assessment with absolute error, relative error.
Illustration	Condition: F1; T5
	Result: T3

Case 2.3 Validation of the Chiller model

Purpose of validation	The influence of the decrease of the flow rate of 1st side to output temperature. To simulate the decrease or leakage of the flow rate of primary coolant.
Time	400 s
Condition of simulation and experiment	Tamb (ambient temperature) : 26 ℃
Disturbance	F1 (flow rate of 1st side) decreasing
Operation in experiment	Maintain F3 (flow rate of 2nd side) stable at 60 l/Min by operating the 3-way valve CV-34 in 2nd side. Maintain T5 (inlet temperature of 1st side) stable at 30 ℃, by setting C2 auto mode and giving setpoint of the outlet temperature T2. Chiller pump is on. Decrease F1 (flow rate of 1st side) from 9 to 4 (l/Min).
Simulation mode	Test mode
Simulation input	F1 (flow rate of 1st side), by using Table Block.
	T5 (inlet temperature of 1st side), by using Table Block.
	F3 (flow rate of 2nd side), preprocessed into smooth data and input by using Table Block.

	T6 (inlet temperature of 2nd side), by using Table Block.
Output for comparison	T3 (outlet temperature of 1st side)
Operation in simulation	Identify the Nusselt number correlation coefficients.
Evaluative method	T3: quantitative assessment with absolute error, relative error.
Illustration	Condition: T5; F1
	Result: T3

Case 2.4 Validation of the Chiller model

Purpose of validation	The influence of the increase of the flow rate of 1st side to output temperature. To simulate the rising of the flow rate of primary coolant.
Time	400 s
Condition of simulation and experiment	Tamb (ambient temperature) : 26 ℃
Disturbance	F1 (flow rate of 1st side) increasing
Operation in experiment	Maintain F3 (flow rate of 2nd side) stable at 60 l/Min by operating the 3-way valve CV-34 in 2nd side. Maintain T5 (inlet temperature of 1st side) stable at 30 ℃, by setting C2 auto mode and giving setpoint of the outlet temperature T2. Chiller pump is on. Increase F1 (flow rate of 1st side) from 4 to 9 (l/Min).
Simulation mode	Test mode
Simulation input	F1 (flow rate of 1st side), by using Table Block.
	T5 (inlet temperature of 1st side), by using Table Block.
	F3 (flow rate of 2nd side), preprocessed into smooth data and input by using Table Block.

	T6 (inlet temperature of 2nd side), by using Table Block.
Output for comparison	T3 (outlet temperature of 1st side)
Operation in simulation	Identify the Nusselt number correlation coefficients.
Evaluative method	T3: quantitative assessment with absolute error, relative error.
Illustration	Condition: T5; F1
	Result: T3

Case 3.1 Validation of the Pressurizer model

Purpose of validation	The response from the air flow rate to the inner pressure. To simulate the increase of vapor to the inner pressure in Pressurizer.
Time	50 s
Condition of simulation and experiment	Tamb (ambient temperature): 23 ℃
	Air supply pressure: 30 PSI
	CV-11 (connect valve to primary loop): 100% (WITH PRESSURIZER mode so as to operate CV-9 manually)
	CV-12 (water drain valve): 0%
	CV-10 (air release valve) : 0%
	L3 (water level): 52%
Operation in experiment	Open CV-9 (air inlet valve) to an appropriate position to increase the inner pressure quickly.
Simulation mode	
Simulation input	Input CV-9 (air inlet valve) position by using Table Block.
Output for comparison	P4 (inner pressure): 0 -> 15 (PSI)
Operation in simulation	Identify appropriate resistances of CV-9 and Regulator.
Evaluative method	P4: quantitative assessment with absolute error
Illustration	Condition: CV-9

	Result: P4, L3
Reference simulation variables	Air density

Case 3.2 Validation of the Pressurizer model

Purpose of validation	The response from the air flow rate to the inner pressure on different water levels. To test the influence from the water level to the inner pressure.
Time	230 s
Condition of simulation and experiment	Tamb (ambient temperature): 23 ℃
	Air supply pressure: 30 PSI
	CV-11 (connect valve to primary loop): 100% (WITH PRESSURIZER mode so as to operate CV-9 manually)
	CV-12 (water drain valve): 0%
	CV-10 (air release valve) : 0%
	L3 (water level): 32%
Operation in experiment	Open CV-9 (air inlet valve) to an appropriate position to increase P4 (inner pressure) slowly.
Simulation mode	
Simulation input	Input CV-9 (air inlet valve) by using Table Block.
Operation in simulation	Identify appropriate resistances of CV-9 and Regulator.
Output for comparison	P4 (inner pressure): 0 -> 15 (PSI)
Evaluative method	P4: quantitative assessment with absolute error
Illustration	Condition: CV-9
	Result: P4, L3
Reference simulation variables	Air density

Case 3.3 Validation of the Pressurizer model

Purpose of validation	The response from the flow rate of released air to the inner pressure. To simulate the release of vapor to the inner pressure in Pressurizer.
Time	100 s
Condition of simulation and experiment	Tamb (ambient temperature): 23 ℃
	CV-11 (connect valve to primary loop): 100% (WITH PRESSURIZER mode so as to operate CV-10 manually)
	CV-12 (water drain valve): 0%
	CV-9 (air inlet valve) : 0%
	L3 (water level): 50%
Operation in experiment	Open CV-10 (air release valve) to an appropriate position to release the inner pressure.
Simulation mode	
Simulation input	Input CV-10 (air release valve) position by using Table Block.
Output for comparison	P4 (inner pressure) :15 ->0 (PSI)
Operation in simulation	Identify appropriate resistance of CV-10
Evaluative method	P4: quantitative assessment with absolute error
Illustration	Condition: CV-10
	Result: P4, L3
Reference simulation variables	Air flow rate out of the tank, water volume, air volume, air density

Case 4.1 Validation of the HX-Tank model

Purpose of validation	The response from the air flow rate to the inner pressure and water level. To simulate the change of vapor to the inner pressure and water level.

Time	180 s
Condition of simulation and experiment	Tamb (ambient temperature): 25 ℃
	Air supply pressure: 30 PSI
	L4 (water level): 25% without bubbles
	CV-5 (air release valve): 20%
Operation in experiment	Open CV-21 (air inlet valve) slowly to raise the inner pressure P4. Water valves CV-18 and CV-25 use auto control to maintain water volume in the tank, better to be closed.
Simulation mode	
Simulation input	Input CV-21 (air inlet valve) by using Table Block.
Operation is simulation	Identify appropriate resistances of CV-5, CV-21 and Regulator. Identify appropriate bubble coefficient to make the water level has similar change between simulation and experiment.
Output for comparison	P2 (inner pressure): 0 -> 15 (PSI)
	L4 (water level)
Evaluative method	Comparison of the results of experiment and simulation
Illustration	Condition: CV-21
	Result: P2, L4
Reference simulation variables	Air density

Case 4.2 Validation of the HX-Tank model

Purpose of validation	The response from the water flow rate to the water level. To simulate the process of filling water to the tank.
Time	300 s
Condition of simulation and experiment	Pump2 (feeding water pump): on
	P2 (inner pressure): 0 PSI (no bubbles)

	L4 (water level): 17%
	CV-18 (water drain valve) : 30%
Operation in experiment	Open CV-25 (water inlet valve): 20% -> 80%
Simulation mode	
Simulation input	Input CV-25 (water inlet valve) with ramp signal.
Operation is simulation	Identify appropriate resistances of CV-25 and CV-18 and linked pipes.
Output for comparison	L4 (water level)
Evaluative method	L4: quantitative assessment with absolute error, relative error.
Illustration	Condition: CV-25
	Result: L4

Case 5.1 Validation of the Turbine model

Purpose of validation	The response from the air flow rate air to the shaft speed of the turbine.
Time	170 s
Condition of simulation and experiment	CV-8 (air outlet valve): 100%
Operation in experiment	Open CV-4 (air inlet valve) from 0 to 100%.
Simulation mode	Test mode
Simulation input	Input F2 (air flow rate) by using Table Block.
Operation in simulation	Identify the moment of inertia, friction factor of the turbine shaft.
Output for comparison	speed (shaft speed)
Evaluative method	speed: quantitative assessment with absolute error,
Illustration	Condition: F2
	Result: speed

Case 5.2 Validation of the Turbine model

Purpose of validation	The relationship of the air inlet valve, the pressure and the flow rate.
Time	90 s
Condition of simulation and experiment	Air supply pressure: 30 PSI
	CV-8 (air outlet valve): 100%
Operation in experiment	Open CV-4 (air inlet valve) from 0 to 100%.
Simulation mode	Normal mode
Simulation input	Input CV-4 (air inlet valve) by using Table Block.
Operation in simulation	Identify the resistance of CV-4, and the resistances of Turbine and CV-8.
Output for comparison	F2 (air flow rate)
	P3 (air pressure after CV-4)
Evaluative method	F2, P3: qualitative assessment
Illustration	Condition: CV-4
	Result: F2, P3

Case 6.1 Validation at system level

Purpose of test	To check the heat transportation between Heater and Chiller, with identifying the appropriate parameters of the thermal inertias between Heater and Chiller.
Loops	Primary coolant loop and secondary water loop, including Heater and Chiller.
Time	120 s
Condition of simulation and experiment	Tamb (ambient temperature) : 25 ℃
	F1 (water flow rate of Heater): 6.2 l/Min
	F3 (water flow rate of 2nd side of Chiller): 20 l/Min

Operation in experiment	Operate C2 (current of Heater): 0->30->0 (A)
Simulation mode	Heater and Chiller: Normal mode
Simulation input	C2 (current of Heater): by using Table Block.
Operation in simulation	Identify the length of the blocks Thermal Heater and Thermal Chiller, which are between Heater and Chiller, named as
Output for comparison	T2 (outlet temperature of Heater) - T5 (inlet temperature of 1st side of Chiller)
	T3 (outlet temperature of 1st side of Chiller) - T1 (inlet temperature of Heater)
Evaluative method	The difference of T2 (outlet temperature of Heater) and T5 (inlet temperature of 1st side of Chiller): qualitative assessment.
	The difference of T3 (outlet temperature of 1st side of Chiller) and T1 (inlet temperature of Heater): qualitative assessment.
Illustration	(T2-T5), (T3-T1)

Case 6.2 Validation at system level

Purpose of test	To test the process of energy conversion and transportation initiated by increasing the current (input power).
Loops	Primary coolant loop and secondary water loop, including Heater and Chiller.
Time	360 s
Condition of simulation and experiment	Tamb (ambient temperature) : 24 ℃
	F1 (flow rate of 1st side of Chiller): 7 l/Min
	F3 (flow rate of 2nd side of Chiller): 28 l/Min
	T6 (inlet temperature of 2nd side of Chiller): 9 ℃
Operation in experiment	Operate C2 (current of Heater): 0 -> 30 ->0 (A)
Simulation mode	Heater and Chiller: Normal mode

Simulation input	C2 (current of Heater): by using Table Block.
Operation in simulation	Check the heat transportation in the closed loop including Heater and Chiller. Identify the parameters of the block Water Cooler to balance the heat production and heat release.
Output for comparison	T3 (outlet temperature of 1st side of Chiller)
Evaluative method	Check the process from C2 (current of Heater) to T3 (outlet temperature of 1st side of Chiller): quantitative assessment with absolute error or relative error.
Illustration	Condition: C2
	Results: T3
Reference simulation variables	Q1 (generated heat in Heater)
	Q2 (released heat to surroundings in Chiller)

Case 6.3 Validation at system level

Purpose of test	The process of using Pressurizer to control the pressure of primary loop.
Loops	Primary coolant loop and air loop, including Heater and Pressurizer.
Time	240 s
Condition of simulation and experiment	Tamb (ambient temperature) : 24 ℃
	Air supply pressure: 30 PSI
	CV-11 (connect valve to primary loop): 100% (WITH PRESSURIZER Mode)
	F1 (flow rate of heater): 6 l/Min
	L3 (water level of Pressurizer): 50%
Operation in experiment	Open CV-9 (inlet valve) and then open CV-10 (outlet valve) to

	increase and then decrease P4 (inner pressure of Pressurizer) and P1 (pressure of primary coolant loop).
Simulation mode	Heater: Normal mode
Simulation input	
Operation in simulation	Open CV-9 (inlet valve) and then CV-10 (outlet valve) to increase and then decrease P4 (inner pressure of Pressurizer) and P1 (pressure of primary coolant loop).
Output for comparison	(P4- P1): the difference of P4 and P1
Evaluative method	Check the pressure difference (P4-P1): quantitative assessment with absolute error or relative error.
Illustration	Condition: P4, P1
	Results: (P4-P1)

Appendix F: User Manual

Version	Release Date	Description	Author
v1.0	August 2015		

Table of Contents

List of Tables

List of Figures

1 Introduction

1.1 Purpose

The purpose of the simulation system is to simulate Nuclear Process Control Test Facility (NPCTF). The model blocks and the simulator system are developed in the environment of Matlab Simulink with Simscape Toolbox.

1.2 Software

The simulator system is built on the platform of Matlab 2013b (8.2.0.701), 32-bit. This version of Matlab must contain Simulink Library and the Simscape Toolbox including Foundation Library as shown as Figure F-1.

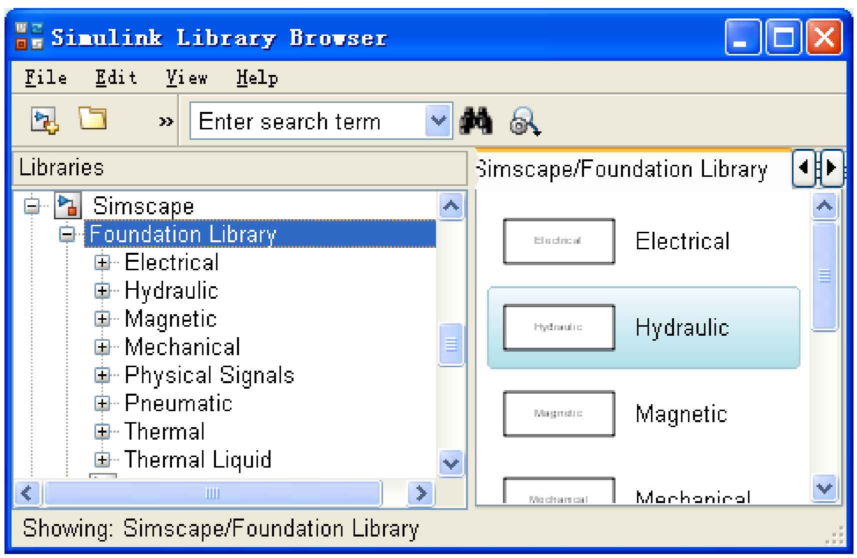


Figure F-1 Simscape Toolbox in Simulink library

The simulation system cannot be executed on an older version of Matlab, and may need to be tested and revised on a new version. The diagram file of the simulation system is named as *pctfModel.slx*, where *slx* is the postfix of Simulink file. The subfolder *+PCTF_BlockLib* contains the files of custom component blocks. This subfolder cannot be renamed when transplanted to a new computer. The file *PCTF_BlockLib_lib.slx*, as shown in Figure F-2, is generated from the block textual files in the subfolder *+PCTF_BlockLib* and can be used as a custom component library.

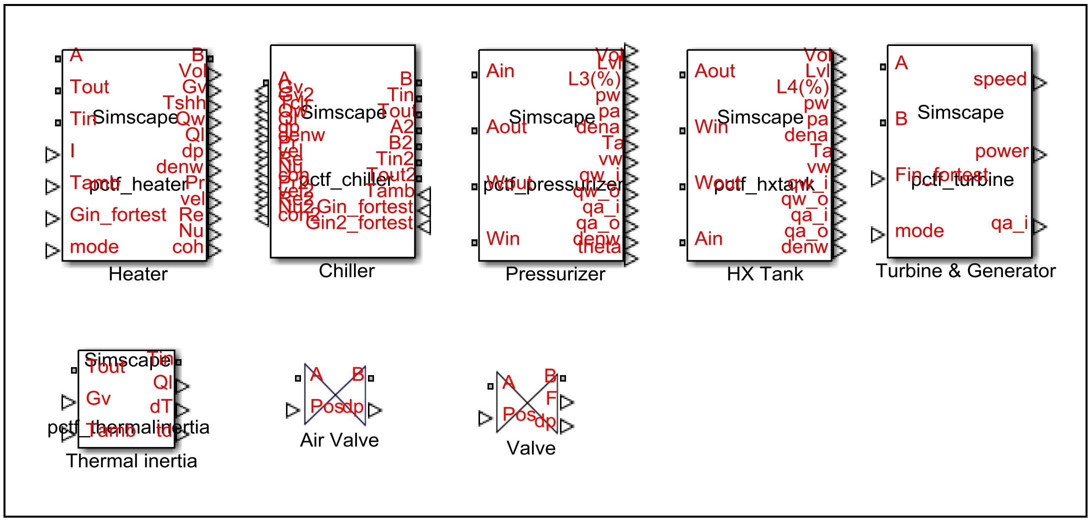


Figure F-2 Custom-defined blocks

1.3 Transplantation

1) Copy the whole folder *pctfModeling* to the destination computer, as an example, to "My Documents."

2) Open Matlab, at *Home menu* click *Set Path*, then click *Add Folder* to add the folder *pctfModeling* into MATLAB searching path.

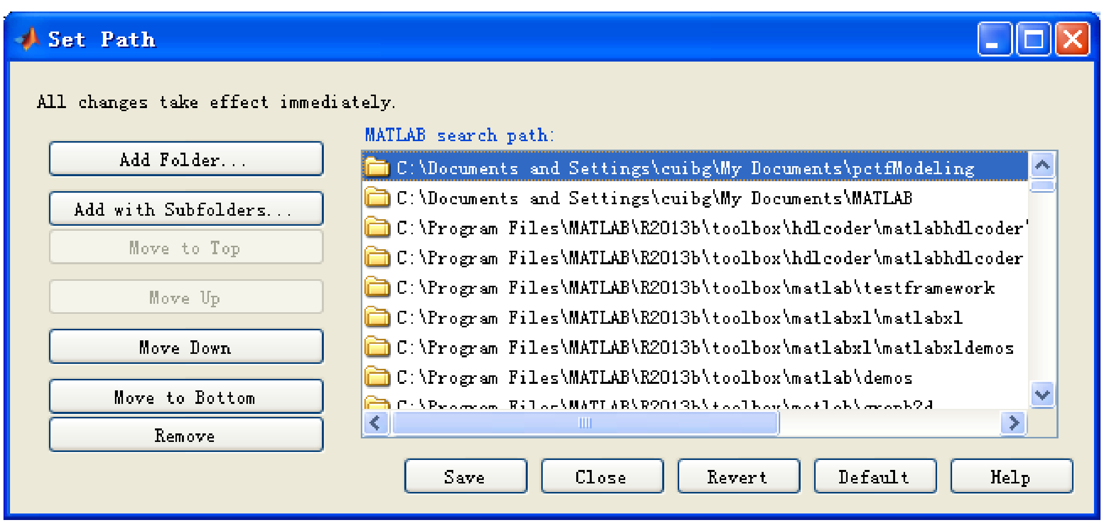


Figure F-3 Matlab searing path

3) In the address bar, find the folder *pctfModeling* and select it as *Current Folder*.

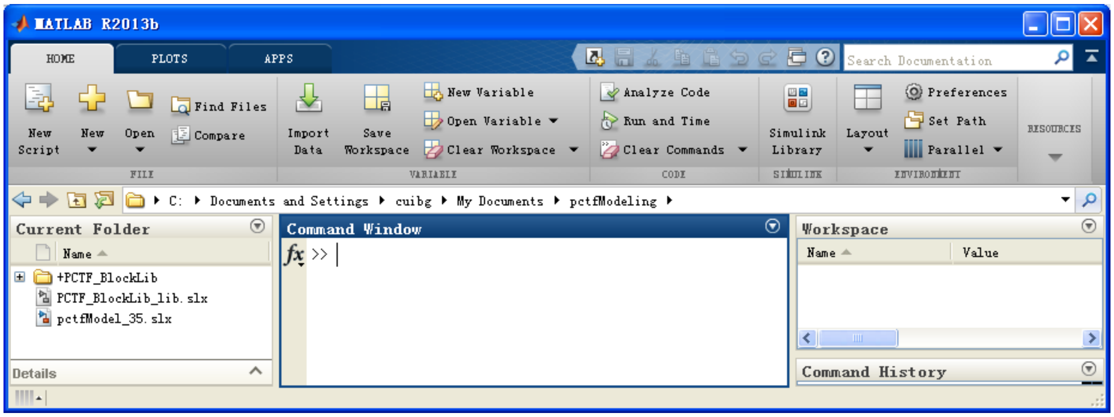


Figure F-4 Selection of current folder

4) Generally, it is unnecessary to rebuild the custom library. If meeting casual event which requires rebuilding it, follow the steps as:

a) Double-click the subfolder *+PCTF_BlockLib* to open it;

b) Input ***ssc_build*** in the Command Window. It may take a few seconds to generate the file *PCTF_BlockLib_lib.slx*.

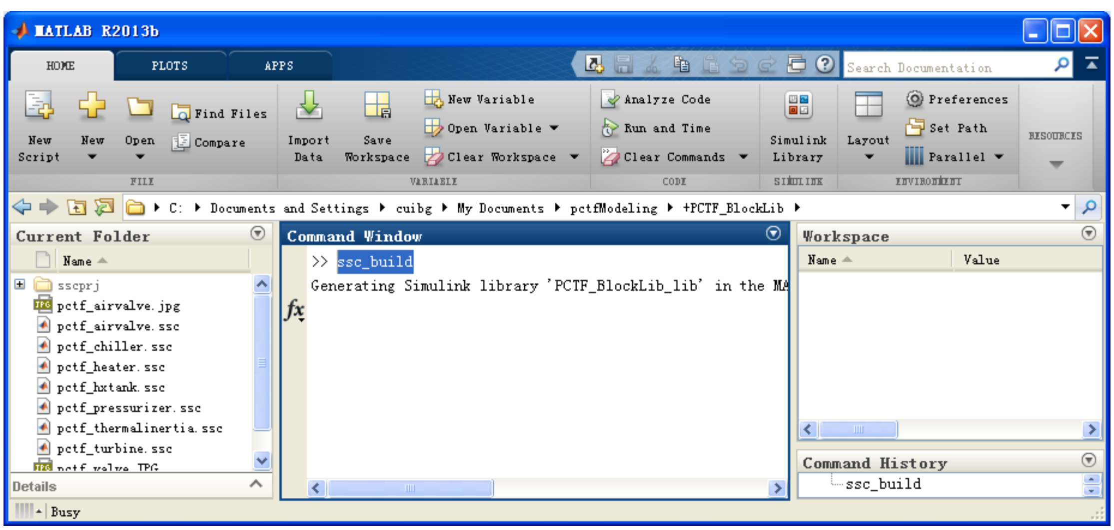


Figure F-5 Block build command

2 Mathematical relations

The built-in block libraries of Simscape cannot provide all the component blocks that used to build the simulator system of NPCTF. Hence, the blocks of the main components and some other general components of NPCTF are developed based on their underlying mathematical relations. All the mathematical equations are expressed as differential equations and algebraic relations, which are derived from conservation laws or constitutive relations. Differential equations, which are mostly as ordinary differential equations (ODEs), represent the calculations of time-dependent variables in dynamic processes. In modeling, lumped parameter models are adopted as the descriptions for the dynamic processes.

Table F-1 Nomenclature

Notation	Description	SI Unit	Conversion
t	calculation step	s	
u	velocity	m/s	
h	convective heater transfer coefficient	$W/(m^2.K)$	
k	thermal conductivity	$W/(m.K)$	
g	gravity acceleration	N/kg or m/s^2	g=9.80665 N/kg
p	pressure	Pa	$1Pa=1.450377\times10^{-4}\ PSI$ 1 feet=0.3048meter $1(feet)=0.3048\times\rho g(Pa)$
Cp	specific heat at constant pressure	$J/(kg.K)$ or $J/(kg.℃)$	
T	thermodynamic temperature	K	
F	mass flow rate	kg/s	
G	volumetric flow rate	m^3/s	
Q	heat transfer rate	J/s	

L	water level	m	
P	power	W	
U	voltage	$V(Volt)$	
I	current	$A(Ampere)$	
V	volume	m^3	$1\ m^3=1000liter$ 1 *US gallon=3.78541 liter*
m	mass	kg	
H	height	m	$1m=39.370in=3.2808ft$
l	length	m	
D	diameter	m	
A	area	m^2	
P_{per}	perimeter	m	
J_ω	moment of inertia	$kg.m^2$	
Nu	Nusselt number		
Re	Reynolds number		
Pr	Prandtl number		
Greek symbols			
ρ	density	kg/m^3	
α	thermal diffusivity	m^2/s	
μ	dynamic viscosity	$kg/(m.s)$ *or* $Pa.s$ *or* $N.s/\ m^2$	
v	kinematic viscosity	m^2/s	$1cSt=1mm^2/s=10^{-6}m^2/s$
ω	angular velocity	rad/s	
Subscripts			
w	water		

m	metal
a	air
e	electricity
i	inlet or inner
o	outlet or outer
l	lost heat, or laminar flow
t	turbulent flow
0	initial condition
1	specially denotes primary side
2	specially denotes secondary side
amb	ambient
htr	heater
clr	chiller
shh	sheath of electrical rod
im	immersed

2.1 Heater

1) Specification

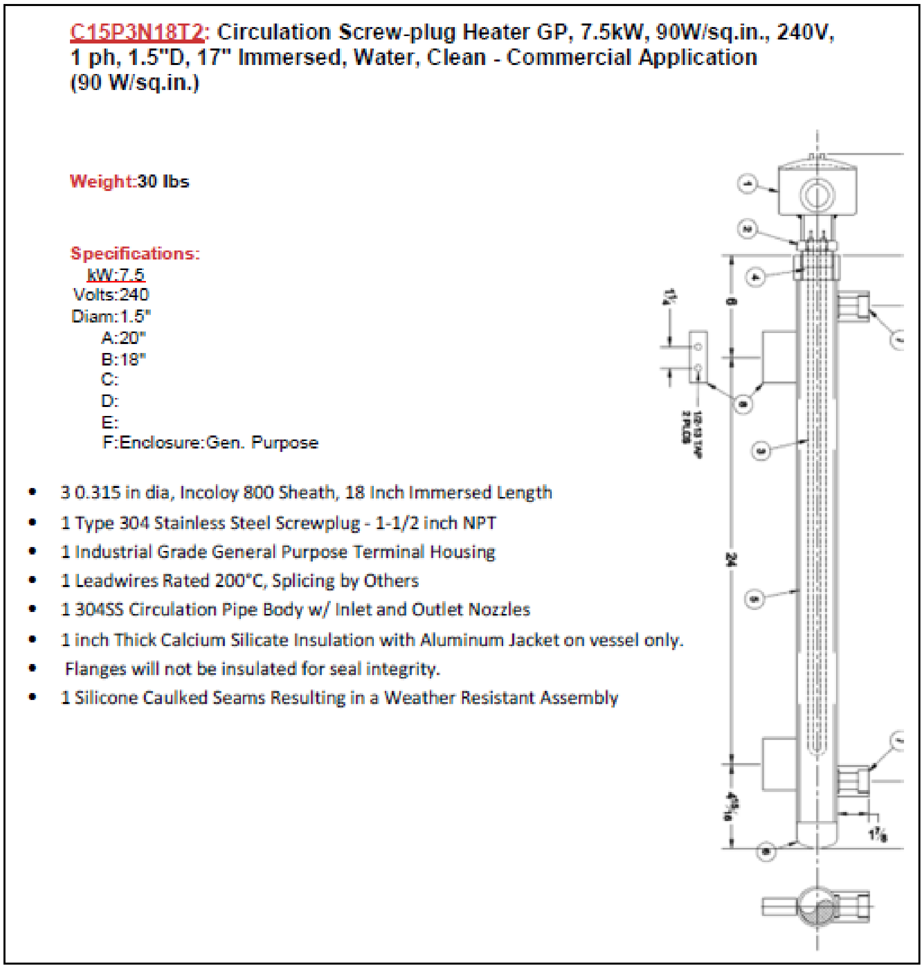


Figure F-6 Specification of the Heater

2) Assumption and simplification

a) It is assumed that the power of electricity is lumped at one point where it is converted to thermal energy and transferred to the metal sheath of the electrical heating rod. Lumped parameter models are employed.

b) Thermal conduction is considered to be purely radial, while axial conduction is neglected due to the high ratio of the length to the diameter of the Heater.

c) The properties of water in term of density, thermal conductivity, specific heat and dynamic viscosity, are assumed to be just temperature dependent under relatively low pressures and low temperatures (<100 ℃).

3) Power from electric to efficient thermal energy:

$$Q_e = \eta P_e = \eta UI \tag{1}$$

where η: the efficiency of energy conversion.

P_e: electrical average power, *J/s (W)* .

To reflect the time required for the energy transferred to the entire sheath wall, the coefficient of time constant τ is added.

$$\tau \frac{dQ_e}{dt} = \eta UI - Q_e \tag{2}$$

4) Heat exchange

Lumped parameter models are employed to calculate the heat exchange.

Water side:

$$\frac{d(\rho_w V Cp_{w,o} T_o)}{dt} = FCp_{w,i}T_i + Q_w - FCp_{w,o}T_o - Q_l \tag{3}$$

Re-written as:

$$(\rho_w V Cp_{w,o}) \frac{d(T_o)}{dt} = G\rho_{w,i}Cp_{w,i}T_i - G\rho_{w,o}Cp_{w,o}T_o + Q_w - Q_l \tag{4}$$

where ρ_w is the average density of water. The inner volume

$$V = \frac{\pi D_{htr}{}^2}{4} H - \frac{\pi D_{shh}{}^2}{4}(2H_{im} + D_{shh}) \tag{5}$$

The heat exchange between water and metal sheath

$$Q_w = hA_{shh}(T_{shh} - T_o) \tag{6}$$

The heat lost to ambience

$$Q_l = \varepsilon\sigma_0 A_{outer}({T_o}^4 - {T_{amb}}^4) \tag{7}$$

where the Stefan-Boltzmann constant $\sigma_0 = 5.67 \times 10^{-8}\, W/(m^2 \cdot K^4)$. ε denotes the coefficient of heat lost to the ambient.

The area of surface:

$$A_{shh} = \pi D_{shh}(2H_{im} + D_{shh}) \tag{8}$$

$$A_{outer} = \pi D_{htr,o} H_{htr} + 2\,\frac{\pi {D_{htr,o}}^2}{4} \tag{9}$$

Thermal balance of the metal sheath of the electrical heating rod:

$$\frac{d((m_{shh}Cp_{shh})T_{shh})}{dt} = Q_e - Q_w$$

$$(m_{shh}Cp_{shh})\frac{d(T_{shh})}{dt} = Q_e - Q_w \tag{10}$$

5) Convection heat transfer coefficient

$$h = \frac{k\,Nu}{D} \tag{11}$$

Prandtl Number $Pr = \frac{\nu}{\alpha} = \frac{\mu\, Cp}{k}$,

where thermal diffusivity $\alpha = \frac{k}{\rho Cp}$, kinematic viscosity $= \frac{\mu}{\rho}$.

Reynolds number $Re = \frac{vD}{\nu} = \frac{vD\rho}{\mu}$, where fluid velocity $v = \frac{G}{A} = \frac{F}{\rho A} = \frac{G}{\frac{\pi D^2}{4}}$.

Hydraulic diameter $D = \frac{4A}{P_{per}} = D_{htr} - 3D_{shh}$

Nusselt number

$$Nu(Re) = \begin{cases} Nu_l = a_l Re^{b_l} Pr^{c_l}, Re \le Re_l \\ Nu_l + (Nu_t - Nu_l)\frac{Re - Re_l}{Re_t - Re_l}, Re_l < Re < Re_t \\ Nu_t = a_t Re^{b_t} Pr^{c_t}, Re \ge Re_t \end{cases} \tag{12}$$

where the subscripts l and t represent the margins of laminar flow and turbulent flow. The correlation coefficients of external flow were slightly modified according to the validations with experiment results. The Nusselt number correlation coefficients are:

i. Laminar regime, $Re < Re_l$ $[a_l\ \ b_l\ \ c_l\ ] = [0.664\ 0.5\ 0.33]$

ii. Turbulent regime, $Re > Re_t$ $[a_t\ \ b_t\ \ c_t] = [1.56\ 0.85\ 0.33]$

iii. Transition regime, $Re_l < Re < Re_t$

6) Properties of water

λ, Cp, ρ, μ are obtained by looking up tables with *tablelookup* function, where they are considered as only the functions of temperatures under relatively low pressures and low temperatures. Kinematic viscosity ν uses the value from the definition of hydraulic domain.

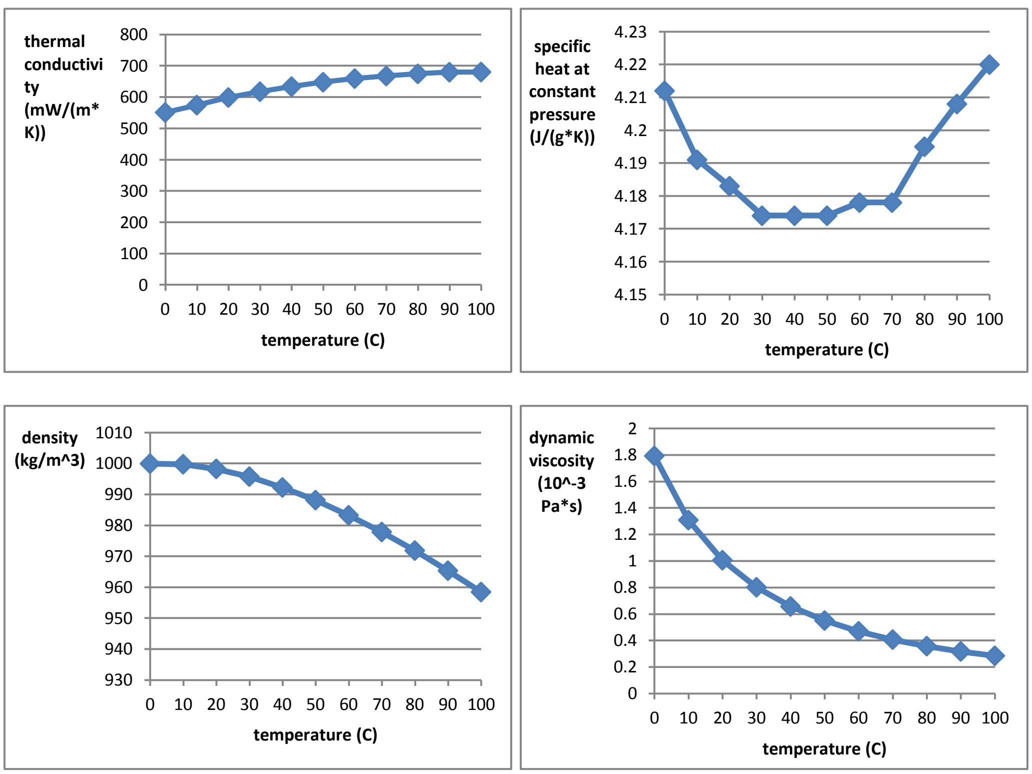


Figure F-7 Properties of water

2.2 Chiller

The Chiller is used to cool the heated water of primary side. There are two sides in the Chiller. The fluid in the primary side is the heated water from the Heater, while the secondary side is linked in the closed cooling circuit in which there is a water cooler. The fluids in both sides exchange heat through a series of metal lamellas.

The mathematical model of the Chiller is separated into primary side and secondary side. Lumped parameter models are employed to compute the heat exchanges.

1) Primary side

$$\frac{d\left((\rho_w V_1 Cp_{w,o})T_o\right)}{dt} = FCp_{w,i}T_i - FCp_{w,o}T_o - Q_1 \tag{13}$$

Rewritten as:

$$(\rho_w V_1 Cp_{w,o})\frac{d(T_o)}{dt} = G\rho_{w,i}Cp_{w,i}T_i - G\rho_{w,o}Cp_{w,o}T_o - Q_1 \tag{14}$$

2) Secondary side

$$\frac{d(\rho_{w,2}V_2 Cp_{w,o,2}T_{o,2})}{dt} = F_2 Cp_{w,i,2}T_{i,2} - F_2 Cp_{w,o,2}T_{o,2} - Q_2$$

$$\rho_{w,2}V_2 Cp_{w,o,2}\frac{dT_{o,2}}{dt} = F_2 Cp_{w,i,2}T_{i,2} - F_2 Cp_{w,o,2}T_{o,2} - Q_2 \tag{15}$$

where the subscript *2* denotes the second side differing from the primary side.

3) Metal lamellas

$$\frac{d((m_{clr}Cp_{clr})T_{clr})}{dt} = Q_1 - Q_2 - Q_l$$

$$m_{clr}Cp_{clr}\frac{dT_{clr}}{dt} = Q_1 - Q_2 - Q_l \tag{16}$$

4) Heat lost to ambience

$$Q_l = \varepsilon\sigma_0 A_{outer}({T_{clr}}^4 - {T_{amb}}^4) \tag{17}$$

where the Stefan-Boltzmann constant $\sigma_0 = 5.67 \times 10^{-8}\ W/(m^2 \cdot K^4)$.ε denotes the coefficient of the heat lost to the ambience.

5) Heat exchange

primary side: $$Q_1 = h_1 A_{inner}(T_{o,1} - T_{clr}) \quad (18)$$

secondary side: $$Q_2 = h_2 A_{inner}(T_{clr} - T_{o,2}) \quad (19)$$

where $T_{\text{o},1}, T_{\text{o},2}$ denote the average temperatures of primary side and secondary side; h_1, h_2 are the convection heat transfer coefficients of primary side and secondary side, which have similar equations as the Heater.

Nusselt number

$$Nu(Re) = \begin{cases} Nu_l = a_l Re^{b_l} Pr^{c_l}, Re \le Re_l \\ Nu_l + (Nu_t - Nu_l)\frac{Re - Re_l}{Re_t - Re_l}, Re_l < Re < Re_t \\ Nu_t = a_t Re^{b_t} Pr^{c_t}, Re \ge Re_t \end{cases} \quad (20)$$

Nusselt Number correlation coefficients are:

i. Laminar regime, $Re < Re_l$ $[a_l\ \ b_l\ \ c_l\ ]$ = [0.664 0.8 0.33]
ii. Turbulent regime, $Re > Re_t$ $[a_t\ \ b_t\ \ c_t]$ = [0.316 0.8 0.33]
iii. Transition regime, $Re_l < Re < Re_t$

6) The properties of water

The same as that described in the section of the Heater.

2.3 Pressurizer

In NPCTF, the Pressurizer is the boundary of air loop and water loop, where the flow rate of air is used to control the pressure.

1) Assumption and simplification

a) Different from the industrial pressurizer with the superheated vapor in the top and the subcooled water in the bottom, the elements in the Pressurizer are only uncompressible water and compressible air.

b) Heat transfer in the vessel, to and from the vessel wall is very small and negligible. The process in the Pressurizer is adiabatic.

c) It is considered that the density of water in the Pressurizer maintain constant due to the assumption of no heat transfer for the Pressurizer. This means the relationship between the mass flow rate and the volumetric flow rate is fixed by water density.

2) Mass balance of water

$$\frac{dV_w}{dt} = G_{w,i} - G_{w,o} \tag{21}$$

where $G_{w,i}$ is the flow rate through CV-11, which may be positive or negative used to maintain the pressure balance in both sides of CV-11. $G_{w,o}$ is the flow rate of drain water to tank. It is assumed the density of water is constant.

3) Mass balance of air

$$\frac{dm_a}{dt} = \frac{d(\rho_a V_a)}{dt} = \Sigma F_{a,i} - \Sigma F_{a,o} \tag{22}$$

4) The density of air

$$\rho_a = \frac{m_a}{V_a} \tag{23}$$

Initially $m_{a,0} = \rho_{a,0} V_{a,0}$

5) Air pressure (absolute pressure) when considering air is ideal gas

$$p_a = \rho_a R_a T_a \tag{24}$$

where the specific gas constant $R_a = \frac{8.314}{28.966} = 287\ J/(kg.K)$.

6) Volume relation

$$V_a + V_w = V = \frac{\pi D^2}{4} l \tag{25}$$

where l is the length the cylinder of the Pressurizer.

7) Water level

It is only considered that the part of the cylinder of the Pressurizer. The bulges of both sides are neglected. The approximation of water level is expressed as the blow.

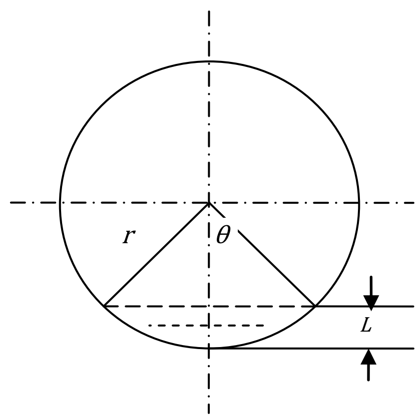


Figure F-8 Water level of the Pressurizer

$$\begin{aligned} &L<r, V_w<\frac{V}{2}, \quad L = r - r\cos\frac{\theta}{2} = r - r\cos\frac{\left(6\times\frac{V_w}{\frac{1}{2}r^2\cdot l}\right)^{\frac{1}{3}}}{2} \\ &L>r, V_a<\frac{V}{2}, \quad L = r + r\cos\frac{\theta}{2} = r + r\cos\frac{\left(6\times\frac{V_a}{\frac{1}{2}r^2\cdot l}\right)^{\frac{1}{3}}}{2} \end{aligned} \tag{26}$$

where the radius $r = \frac{D}{2}$.

8) The pressure at the bottom

$$p = p_w + p_a \tag{27}$$

where the pressure of water $p_w = \rho_w g L_w$.

9) Initial value

Initial air temperature $T_{a,0}$ should be given so as to get initial value of air mass by

$$m_{a,0} = \rho_{a,0}V_{a,0} = \frac{101325}{R_a T_{a,0}}(\frac{\pi D^2}{4}H - V_{w,0}) \tag{28}$$

Atmospheric pressure is considered as *101325 Pa*.

2.4 HX-Tank

The HX-Tank is a pressurized chamber with air and water. When it is charged into water and air, the pressures of water at the bottom and air at upper space are produced. It is assumed water is incompressible.

1) Mass balance of water

$$\frac{dV_w}{dt} = \Sigma G_{w,i} - \Sigma G_{w,o} \tag{29}$$

2) Mass balance of air

$$\frac{dm_a}{dt} = \Sigma F_{a,i} - \Sigma F_{a,o} \tag{30}$$

3) Density of air

$$\rho_a = \frac{m_a}{V_a} \tag{31}$$

Initially $m_{a,0} = \rho_{a,0}V_{a,0}$

4) Air pressure when considering air is ideal gas

$$p_a = \rho_a R_a T_a \tag{32}$$

where the specific gas constant $R_a = 287\ J/(kg.K)$.

5) Volume relation of air and water

$$V_a + V_w = V = \frac{\pi D^2}{4}H \tag{33}$$

6) Water level

$$L_w = \frac{V_w + \xi\ \Sigma F_{a,i}/{\rho_a}^4}{\frac{\pi D^2}{4}} \tag{34}$$

where ξ is set as the bubble coefficient. Initially, $V_{w,0} = \frac{\pi D^2}{4} L_{w,0}$

7) The pressure at the bottom

$$p = p_w + p_a \tag{35}$$

where the pressure of water $p_w = \rho_w g L_w = \rho_w g \frac{V_w}{\frac{\pi D^2}{4}}$

8) Initial value

Initial air temperature $T_{a,0}$ should be given so as to get initial value of air mass by

$$m_{a,0} = \rho_{a,0} V_{a,0} = \frac{101325}{R_a T_{a,0}} \left(\frac{\pi D^2}{4} H - V_{w,0}\right) \tag{36}$$

Atmospheric pressure is considered as *101325 Pa*.

2.5 Turbine

The Turbine model is designed as one simple model that based on the relations of parameters rather than the principles.

1) From Newton's Law, the differential equation for shaft speed ω is defined as

$$J_\omega \frac{d\omega}{dt} = \zeta \Sigma F_a - f\omega \tag{37}$$

where

ζ: the conversion rate from kinetic momentum of air flow, regarding $\xi\ \Sigma F_a$ as the torque that drives turbine running.

J_ω: the moment of inertia of turbine, *kg.m*^{2}

f: viscous friction, *kg.m*^{2}; and $f\omega$ represents the power output or the shaft work of the Turbine.

The units need to be uniformed when programming the equations.

2) Power generated

$$P_o = k \cdot f\omega \tag{38}$$

where k denotes the coefficient of energy conversion from kinetic to electricity, $0 < k < 1$. Kinetic energy is from the air into turbine and transferred to the generator by shaft works.

2.6 Thermal Inertia

This model is used to simulate the delay of heat transport processes. For a definite volume such as a pipe, the change of outlet temperature is determined not only by the inlet temperature, but also by the inner volume.

$$(\rho_w V Cp_{w,o}) \frac{dT_o}{dt} = G\rho_{w,i} Cp_{w,i} T_i - G\rho_{w,o} Cp_{w,o} T_o - Q_l \tag{39}$$

where the heat lost to ambience:

$$Q_l = \epsilon \sigma_0 A_{outer} (T^4 - {T_{amb}}^4) \tag{40}$$

2.7 Water Valve

Since the codes of the built-in components such as pipe, pump, and valve are not reached, and the built-in valve cannot work in this model through test, water valve needs to be developed as a custom component. It is designed as a hydraulic branch with some hydraulic resistance. The actual hydraulic resistance is determined by the through-area. The through-area is calculated from the valve position. The relationship of the through-area and the position is given by the valve characteristic curve inputted by users. The default curve is linear.

The signal port of valve position should be connecting to a port of another block. Limited by Simulink environment, the actual hydraulic resistance of water valve cannot be infinite when receiving a zero position command. In another word, even the position command is full-closed (zero position), the actual hydraulic resistance cannot be computed as a too large value. Otherwise there will be a system error when running the model. Alternatively, in order to simulate this situation, the water valve can be set to full-closed with giving a minimum flow rate like zero in its parameter window. In this way the flow rate is equal to the minimum value directly with disregarding the hydraulic resistance. In the simulator, however, not all water valves can be set full-closed with zero flow rates. This depends on the position of the valve in the system.

2.8 Air Valve

Air valve is designed almost the same as water valve. The description of the water valve can be referenced.

3 Component blocks

Simulink blocks represent basic mathematical operations. Through connecting the ports of blocks, the resulting diagram is equivalent to a mathematical model. Simscape blocks have two types of ports: conserving ports (symbol circle) and signal ports (symbol triangle).

3.1 Heater

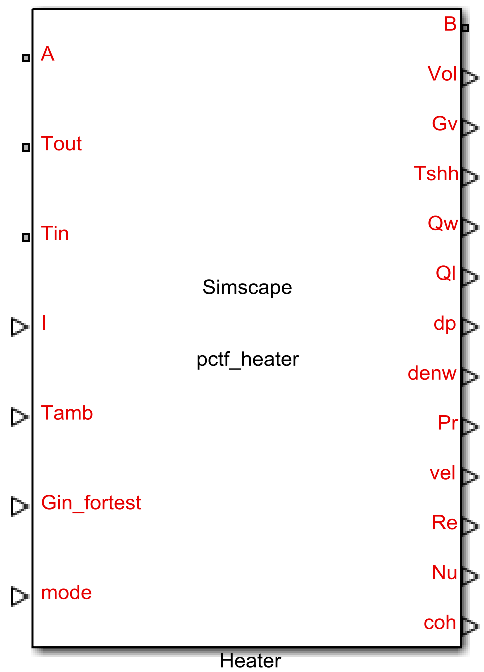


Figure F-9 Heater block

Table F-2 Port of the Heater

denotation	description	default unit
A, B	conserving hydraulic ports, associated with the block inlet and outlet, respectively. The block positive direction is from port A to port B. This means that the flow rate is positive if fluid flows from A to B, and the pressure loss is determined as dp = p_A - p_B.	
Tin, Tout	conserving thermal ports, used to connect inlet and outlet water temperatures.	
I	connecting to current signal which controls the power of the heater	*A*
Tamb	the input of ambient temperature	*K*

Gin_fortest	just for test mode, in which the flow rate = the input	*m^3/s*
mode	test mode switch, 0: normal; 1: test mode	
Vol	inner volume of heater	*m^3*
Gv	water flow rate	*m^3/s*
Tshh	the temperature of metal sheath	*K*
Qw	heat transfer flux between water and sheath	*J/s*
Ql	heat lost to ambient	*J/s*
dp	pressure difference between inlet and outlet	*Pa*
denw	density of water	*kg/m^3*
Pr	Prandtl number	
vel	velocity of water	*m/s*
Re	Reynolds number	
Nu	Nusselt number	
coh	convective heat transfer coefficient	*W/(m^2*K)*

3.2 Chiller

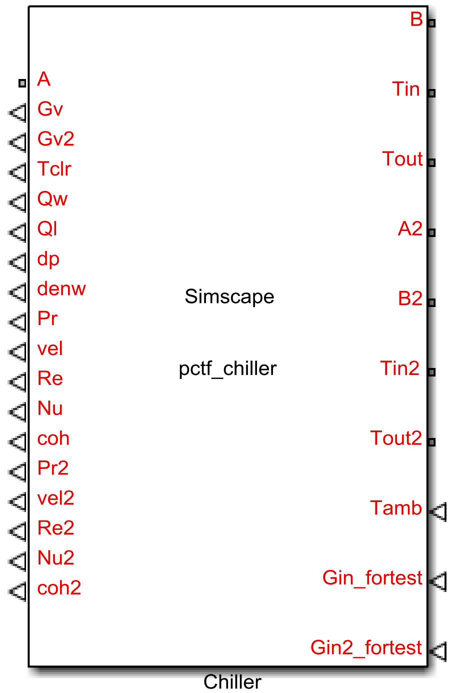


Figure F-10 Chiller block

Table F-3 Port of the Chiller

denotation	description	default unit
A, B	conserving hydraulic ports, used to connect to primary water loop as inlet and outlet. The flow rate is positive if fluid flows from A to B.	
A2, B2	conserving hydraulic ports, used to connect to secondary side water loop as inlet and outlet.	
Tin, Tout	conserving thermal ports, used to connect inlet and outlet water temperatures of primary water loop.	
Tin2, Tout2	conserving thermal ports, used to connect inlet and outlet water temperatures of secondary side water loop.	

Tamb	the input of ambient temperature	*K*
Gin_fortest	just for test mode, in which the flow rate of primary side = the input	*m^3/s*
Gin2_fortest	just for test mode, in which the flow rate of secondary side = the input	*m^3/s*
Gv	water flow rate of primary side	*m^3/s*
Gv2	water flow rate of secondary side	*m^3/s*
Tclr	the temperature of metal lamellas	*K*
Qw	heat transfer flux of primary side	*J/s*
Ql	heat lost to ambient	*J/s*
dp	pressure difference between inlet and outlet of primary side	*Pa*
denw	density of water	*kg/m^3*
Pr	Prandtl number of primary side	
vel	velocity of water of primary side	*m/s*
Re	Reynolds number of primary side	
Nu	Nusselt number of primary side	
coh	convective heat transfer coefficient of primary side	*W/(m^2*K)*
Pr2	Prandtl number of secondary side	
vel2	velocity of water of secondary side	*m/s*
Re2	Reynolds number of secondary side	
Nu2	Nusselt number of secondary side	
coh2	convective heat transfer coefficient of secondary side	*W/(m^2*K)*

3.3 Pressurizer

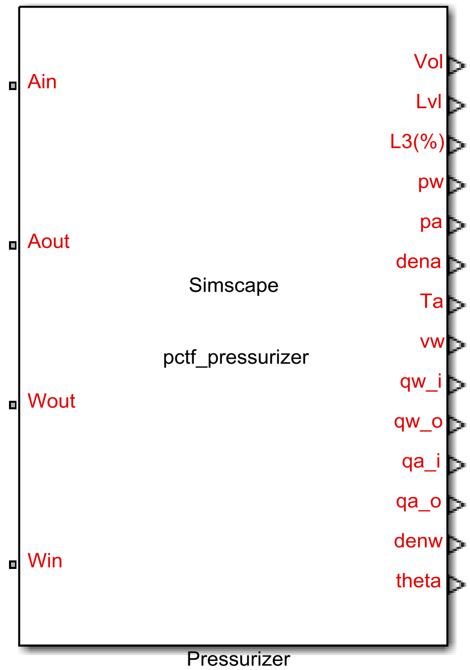


Figure F-11 Pressurizer block

Table F-4 Port of the Pressurizer

denotation	description	default unit
Ain, Aout	conserving pneumatic ports as air input and output (spill)	
Win, Wout	conserving hydraulic ports as water input and output.	
Vol	inner volume of pressurizer	*m^3*
Lvl	water level	*mm*
L3	water level	*%*
pw	gage pressure of water	*Pa*
pa	gage pressure of air	*Pa*

dena	density of air	*kg/m^3*
Ta	air temperature	*K*
vw	volume of water	*m^3*
qw_i	flow rate of water in	*m^3/s*
qw_o	flow rate of water out	*m^3/s*
qa_i	flow rate of air in	*kg/s*
qa_o	flow rate of air out	*kg/s*
denw	density of water	*kg/m^3*
theta	theta of water-covered	

3.4 HX-Tank

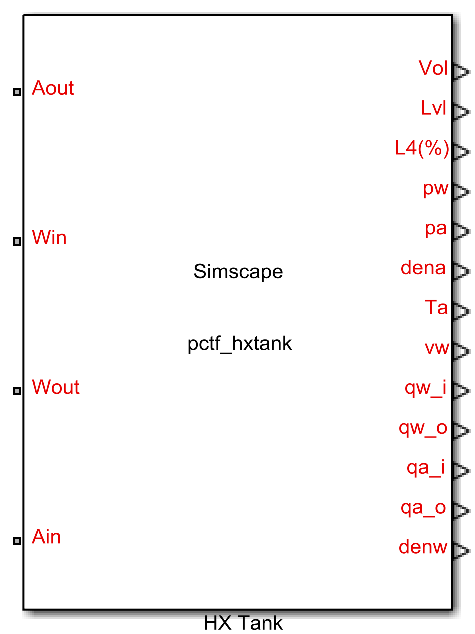


Figure F-12 HX-Tank block

Table F-5 Port of the HX-Tank

denotation	description	default unit

Ain, Aout	conserving pneumatic ports as air input and output (spill)	
Win, Wout	conserving hydraulic ports as water input and output.	
Vol	inner volume of pressurizer	*m^3*
Lvl	water level	*mm*
L4	water level	*%*
pw	pressure of water	*Pa*
pa	pressure of air	*Pa*
dena	density of air	*kg/m^3*
Ta	air temperature	*K*
vw	volume of water	*m^3*
qw_i	flow rate of water into tank	*m^3/s*
qw_o	flow rate of water out of tank	*m^3/s*
qa_i	flow rate of air into tank	*kg/s*
qa_o	flow rate of air out of tank	*kg/s*
denw	density od water	*kg/m^3*

3.5 Turbine

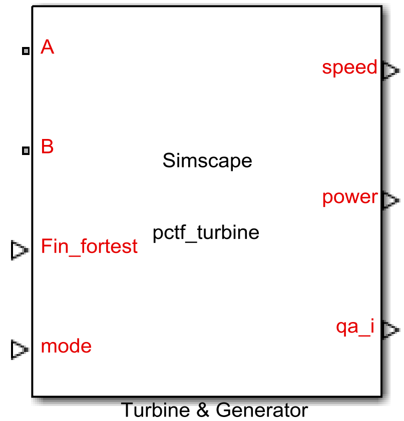


Figure F-13 Turbine block

Table F-6 Port of the Turbine

denotation	description	default unit
A, B	conserving pneumatic ports, used to connect to pressurized air loop as inlet and outlet. The flow rate is positive if fluid flows from A to B.	
Fin_fortest	just for test mode, the air flow rate = the input	*kg/s*
mode	test mode swicth, 0: normal; 1: test mode	
speed	speed of turbine	*rpm*
power	power of generator	*W*
qa_i	flow rate of air through turbine	*kg/s*

3.6 Thermal Inertia

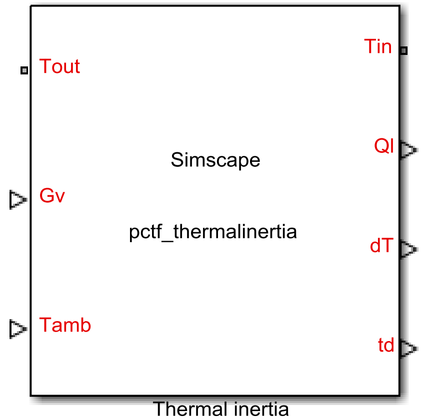


Figure F-14 Thermal Inertia block

Table F-7 Port of Thermal Inertia

denotation	description	default unit
Tin, Tout	conserving thermal ports, used to connect inlet and outlet water temperatures of water loop.	
Gv	water flow rate	*m^3/s*
Tamb	the input of ambient temperature	*K*
Ql	heat lost to ambient	*J/s*
dT	temperature difference between inlet and outlet	*K*
td	time delay	*s*

3.7 Water Valve

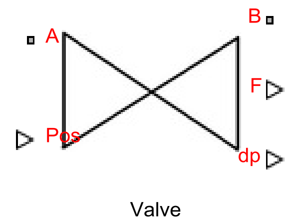


Figure F-15 Water Valve block

Table F-8 Port of Water Valve

denotation	description	default unit
A, B	conserving hydraulic ports, associated with the block inlet and outlet, respectively. The block positive direction is from port A to port B. This means that the flow rate is positive if fluid flows from A to B, and the pressure loss is determined as dp = p_A - p_B.	
Pos	the input of open position signal in uniform	
F	water flow rate	*kg/s*
dp	pressure difference between inlet and outlet	*Pa*

3.8 Air Valve

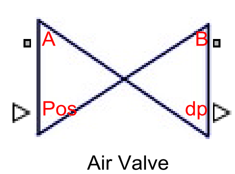


Figure F-16 Air Valve block

Table F-9 Port of Air Valve

denotation	description	default unit
A, *B*	conserving pneumatic ports as air inlet and outlet, The flow rate is positive if fluid flows from A to B.	
Pos	the input of open position signal in uniform	
dp	pressure difference between inlet and outlet	*Pa*

4 Simulator system

4.1 Overview

In the Simulink window, one can press SPACE button to make the overview fit in a view.

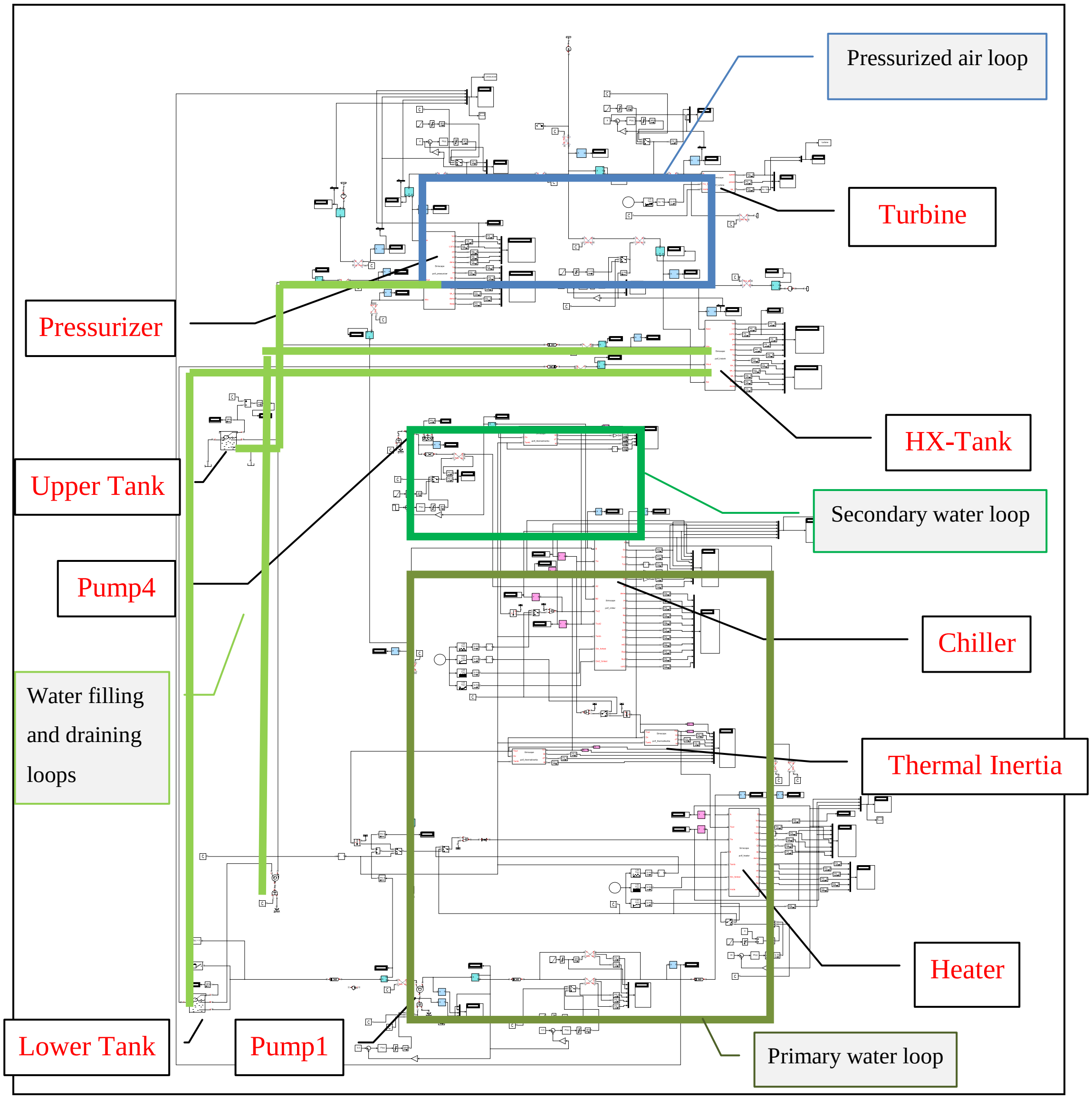


Figure F-17 Overview of the simulator diagram

4.2 Solver configuration block

For a diagram of model system in one page, there must be one Solver Configuration block associated with local solver configuration, as shown in the below figure.

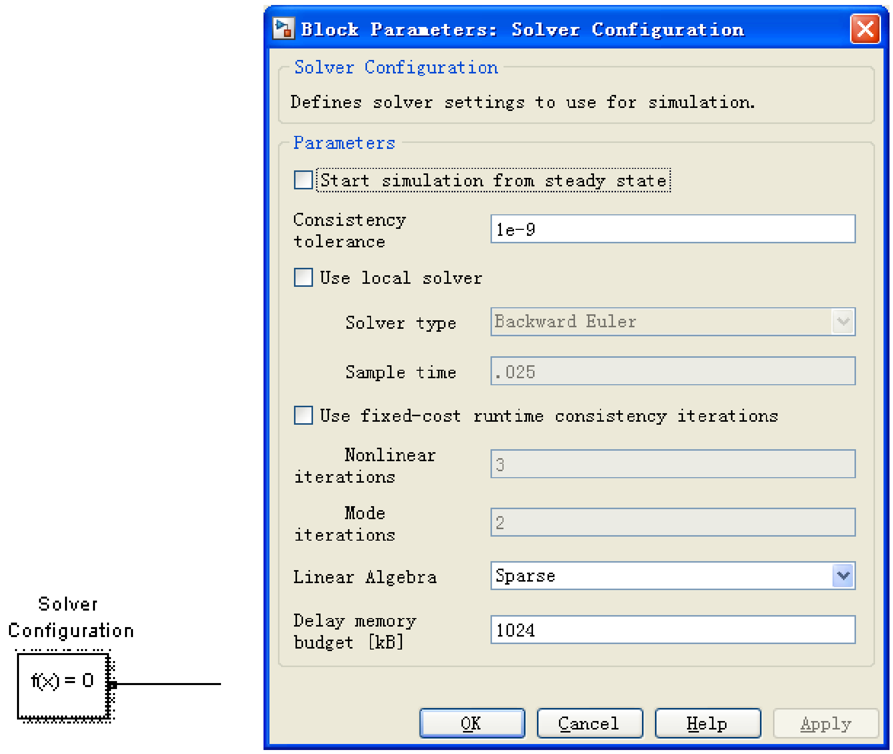


Figure F-18 Solver configuration

4.3 Custom Hydraulic Fluid block

For all the hydraulic loops, there must be only one Custom Hydraulic Fluid block to set fluid properties for the linked loop.

In the hydraulic loop, the fluid properties, such as density and kinematic viscosity, are used in the calculations of all the components from the built-in libraries except the custom components.

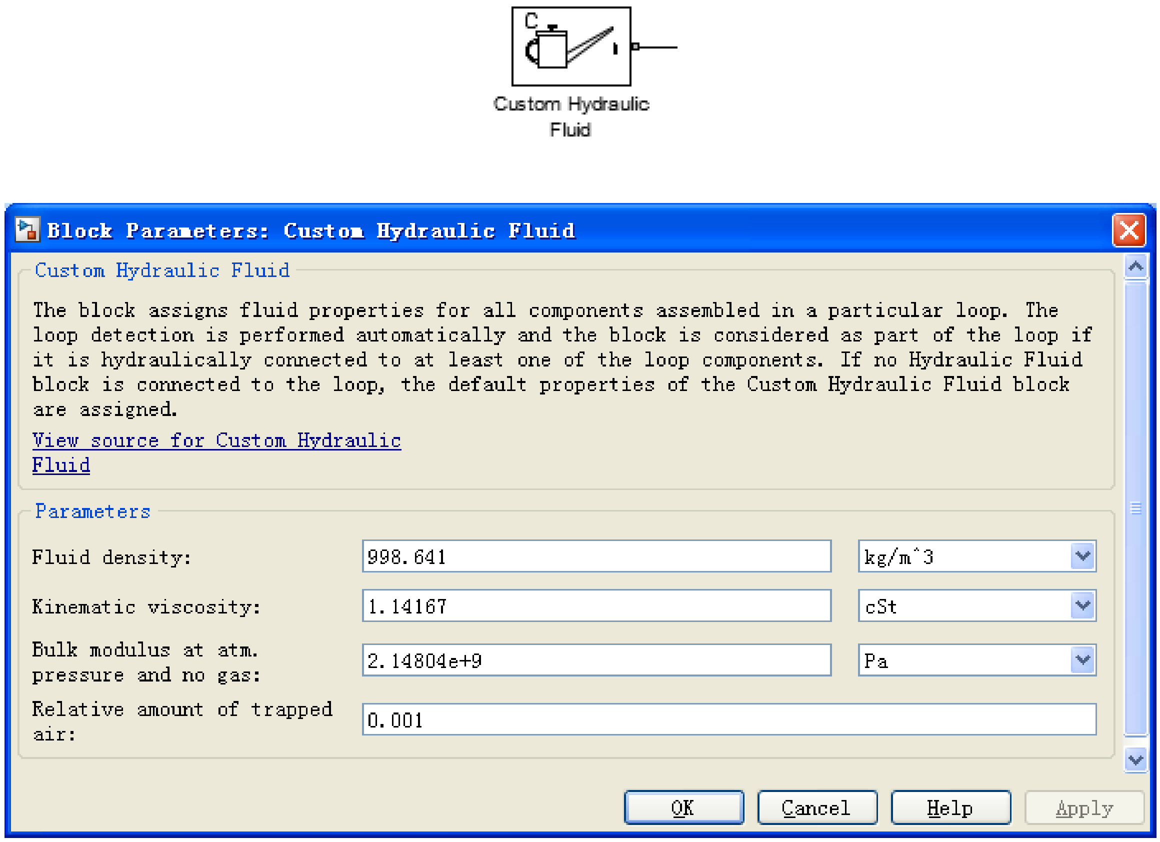


Figure F-19 Hydraulic fluid

4.4 Gas properties block

Like the above hydraulic property block, a Gas Properties Block connecting to gas loops is needed to define air properties, as shown in the below figure. The ambient pressure and the ambient temperature should be given with the correct values, which are used in the air loop. The ambient temperature is transmitted in the air loops.

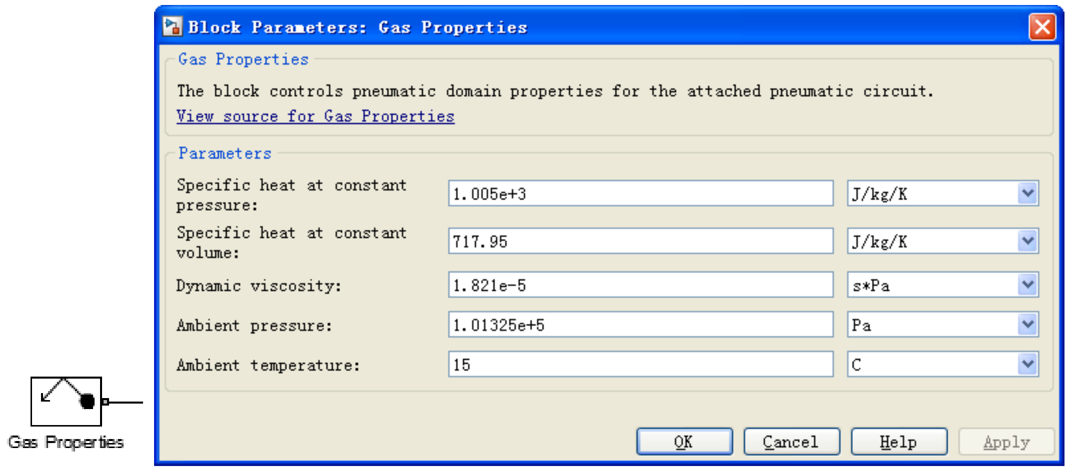


Figure F-20 Gas properties

4.5 Diagram of water loops

The hydraulic blocks and thermal blocks used in the water loops are presented in this section. The guides of those I&C blocks and math blocks can be referenced in Simulink User Manual.

4.5.1 Sensor blocks

1) Flow meter of water

It is used to measure flow rate of water. The measured flow rate is positive if fluids flowing from A to B, otherwise it is negative. The output Q can be linked to a Display block.

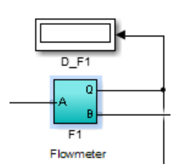


Figure F-21 Flow meter of water

The subsystem can be opened by double-clicking the block. The block Hydraulic Flow Rate Sensor is from Simscape library. The unit of the output Q can be selected in PS-S block, the calculation of converting units is done by the platform.

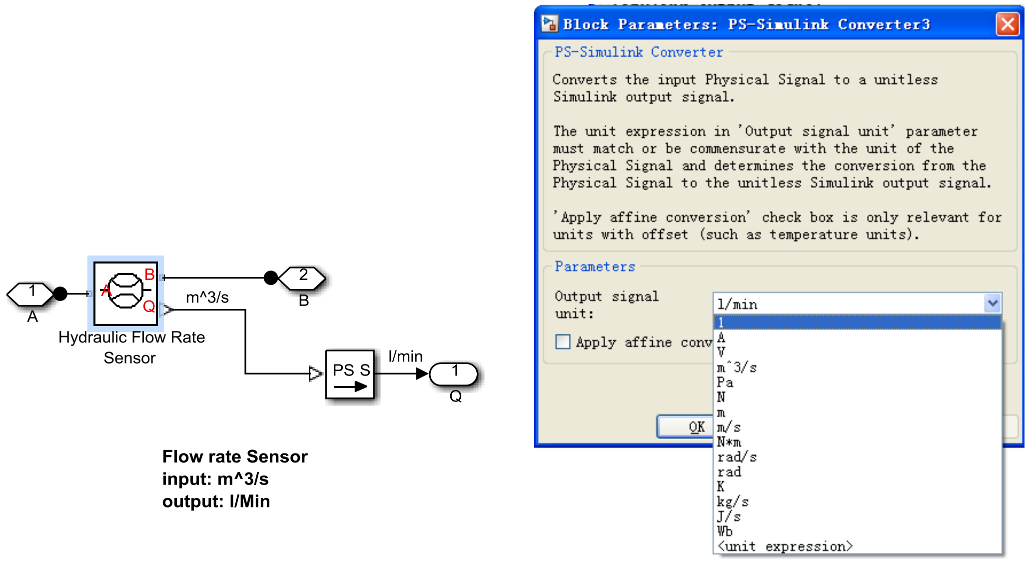


Figure F-22 Output of hydraulic flow rate sensor

2) Pressure sensor of water

It is used to measure pressure of water by attaching port A to the measuring point. The output P can be linked to a Display block for displaying the value in running process.

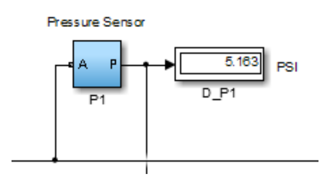
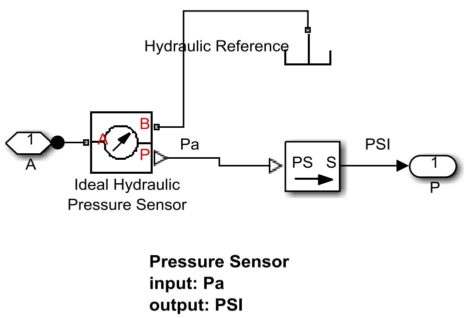


Figure F-23 Pressure sensor of water

The subsystem can be opened by double-clicking the block. The blocks of Ideal Hydraulic Pressure Sensor and Hydraulic Reference are both from Simscape library. Port B of Ideal Hydraulic Pressure Sensor must be connected to a hydraulic port like Hydraulic Reference block. The unit of the output P can be selected in PS-S block, the calculation of converting units is done by the platform.

Hydraulic Reference block defines a *0 Pa* port. Because pumps employ gage pressure in their characteristic curves, the whole water loops use gage pressure accordingly. Therefore, the output of pressure sensors is gage pressure.



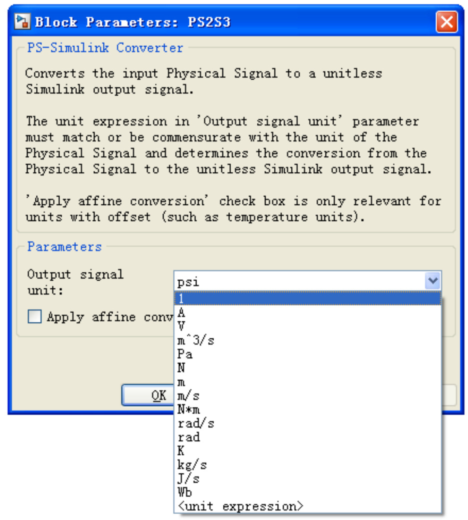


Figure F-24 Output of hydraulic pressure sensor

3) Temperature sensor of water

It is used to monitor the temperature of the attaching point by port A. Port A is conserving thermal port. It must be connected another thermal port.

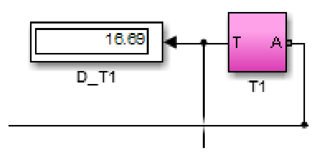


Figure F-25 Temperature sensor of water

The subsystem can be opened by double-clicking the block. The blocks of Ideal Temperature Sensor and Thermal Reference are both from Simscape library. Another embedded subsystem block "Tout" can also be opened. Thermal Reference block represents a thermal point with 0 *K*. Then it is changed to degree Celsius by the "Tout" subsystem block, in which the unit of temperatures changes from Kelvin to Celsius by subtracting *273.15*. The PS-S block is only for export the value to a Simulink block, which output signal unit chooses 1.

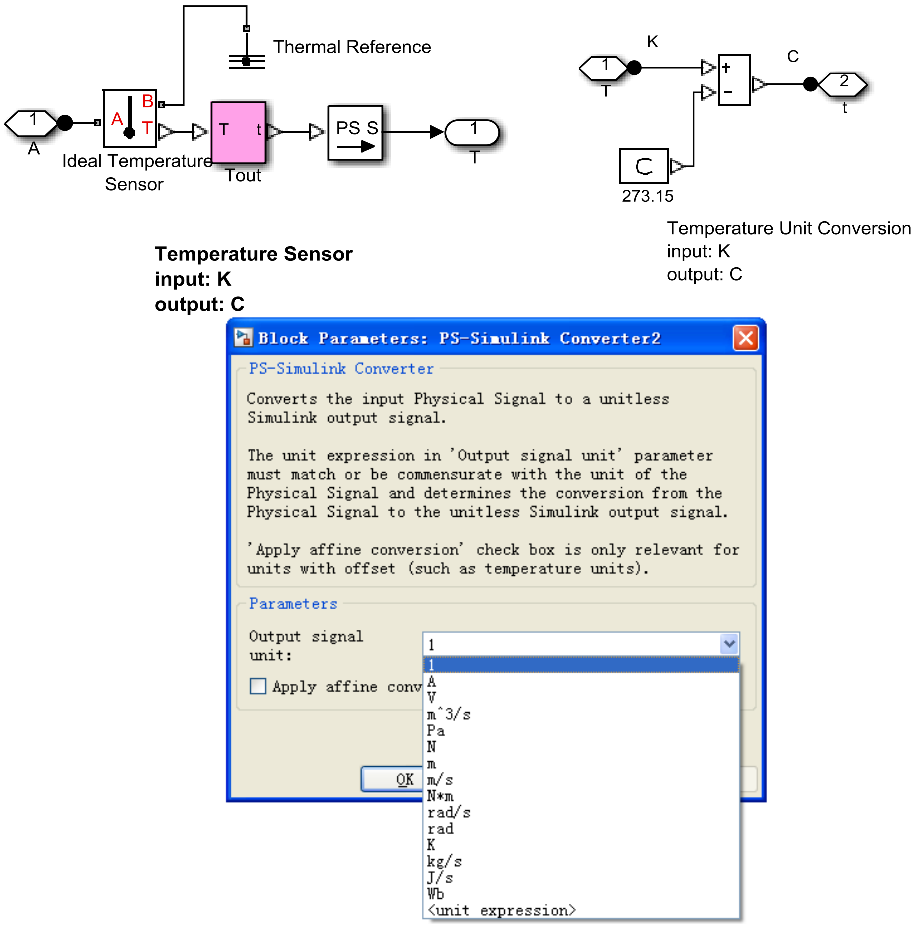


Figure F-26 Output of temperature sensor

4.5.2 Component blocks from Simscape library.

1) Hydraulic Resistance Tube

The block is from the built-in library of Simscape as a resistive component. The source code is open and can be studied with its manual. The following figure represents one application in the secondary water loop.

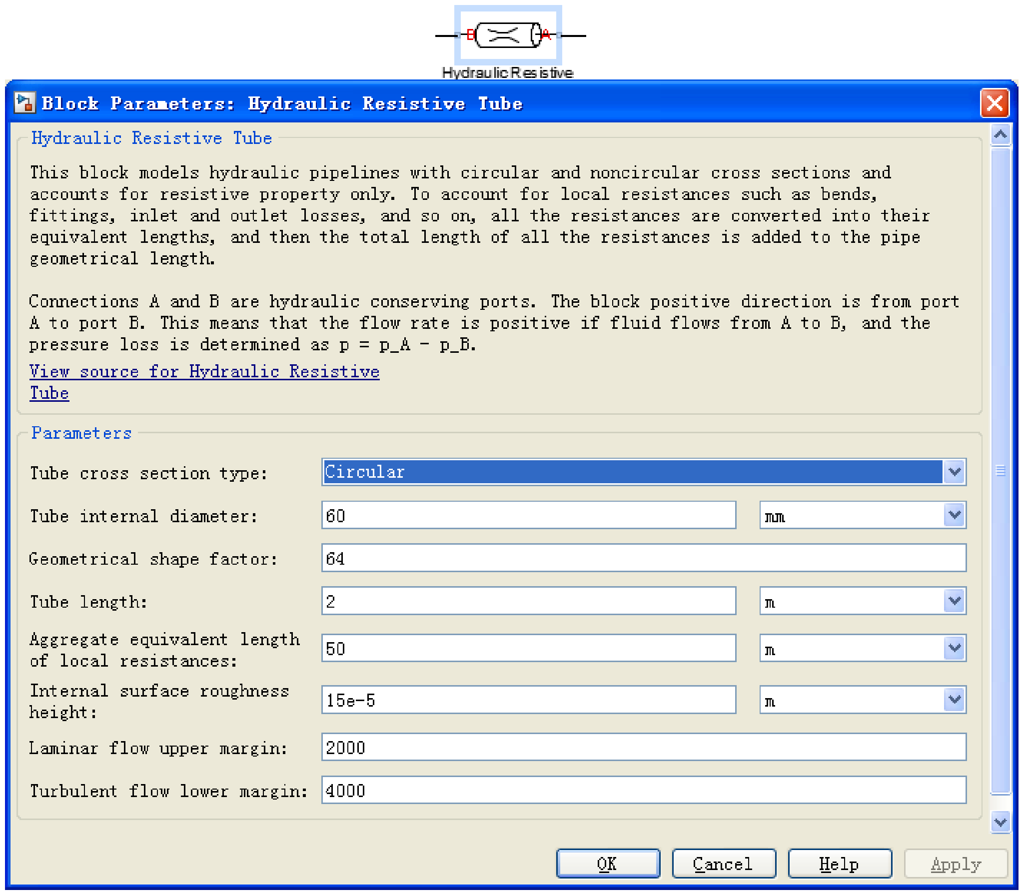


Figure F-27 Hydraulic Resistance Tube

2) Pump1

There are three pumps in the water loop: main pump (Pump1), feeding pump (Pump2), and chilling pump (Pump4). They all use Lookup Table from Simscape library to input their working curve.

Pump1 is the main pump of the primary water loop. Its purpose is to fill water into this loop and then form water circuit in this loop. The specification of Pump1 provides the performance curve of P-Q (pressure-quantity). By converting the units from *GPM-Feet* to *l/Min-PSI*, the dilivery vector data for P-Q is obtained and applied into the block parameter dialogbox. (*GPM: gallon per minute*)

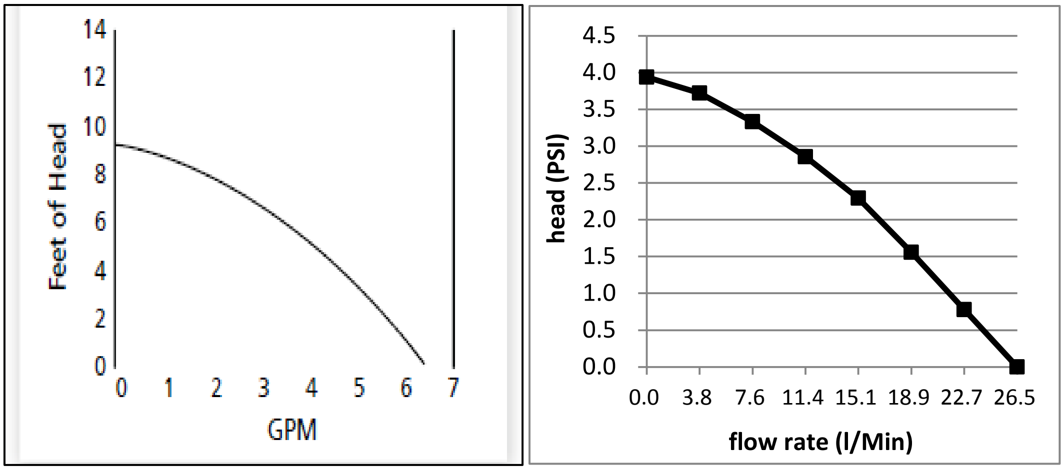


Figure F-28 Pump1

3) Pump2

Pump2 is used to feed water into the HX-Tank. Its performance curve is obtained from its specification.

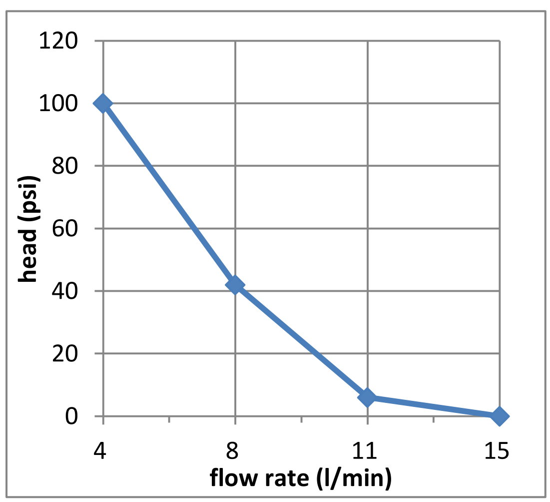


Figure F-29 Pump2

4) Pump4

Pump4 is the chilling pump in the secondary water loop. Its performance curve is also obtained from its specification by converting units.

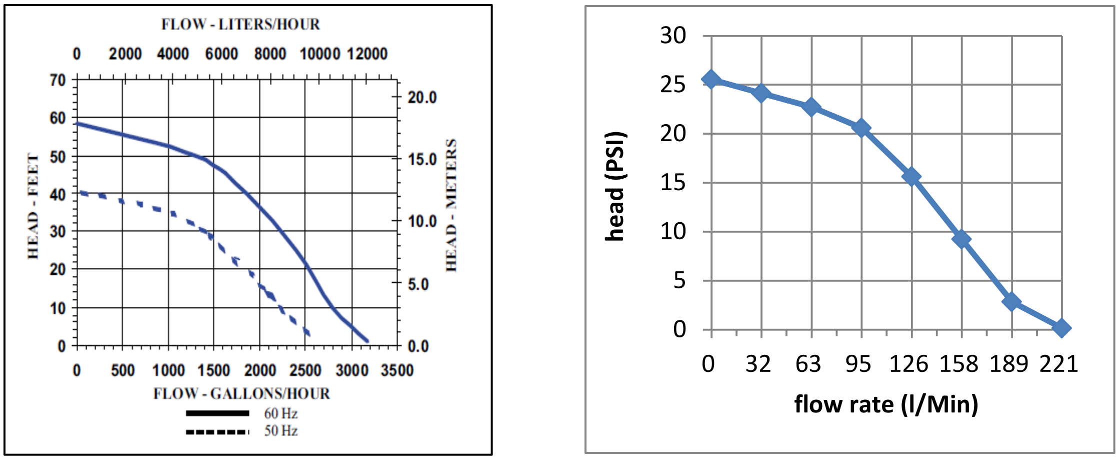


Figure F-30 Pump4

4.5.3 Main components from the custom block library

1) Heater

The Heater subsystem is composed of the Heater block and related input and output blocks. Besides the lines connecting to other component blocks by conserving ports (symbol circle), there are several blocks to input and output variables by signal ports (symbol triangle).

In the Heater block, the conserving hydraulic ports “A” and “B” are connecting to the pipe and the valve in the primary water loop. This connecting loop can form a physical network in the hydraulic domain. The pressure and flow rate are calculated by solving the equation matrix of this network.

The conserving thermal ports “Tin” and “Tout” are used to connect to the Chiller block through thermal inertia blocks. These ports work in the thermal domain, where the connections just transmit the values of the temperatures. Also, in the thermal domain, there are no physical networks as the hydraulic domain and the pneumatic domain.

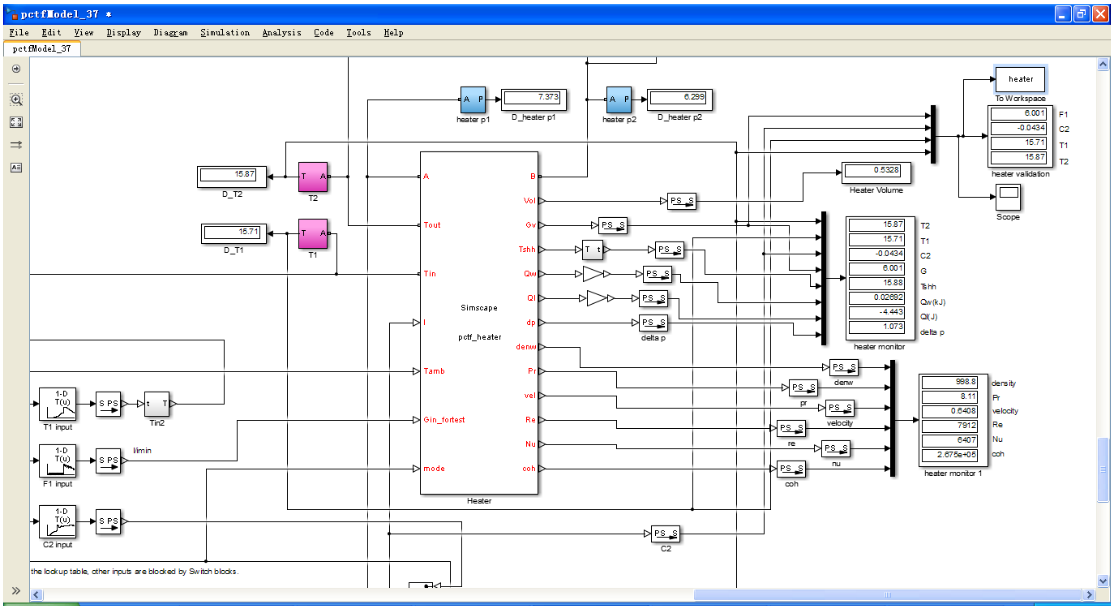


Figure F-31 Diagram of the Heater

The output variable ports need to link to the PS-Simulink Converter to convert physical signal to unitless signal. Commensurate values can be converted in different units by selecting “Output signal unit.”

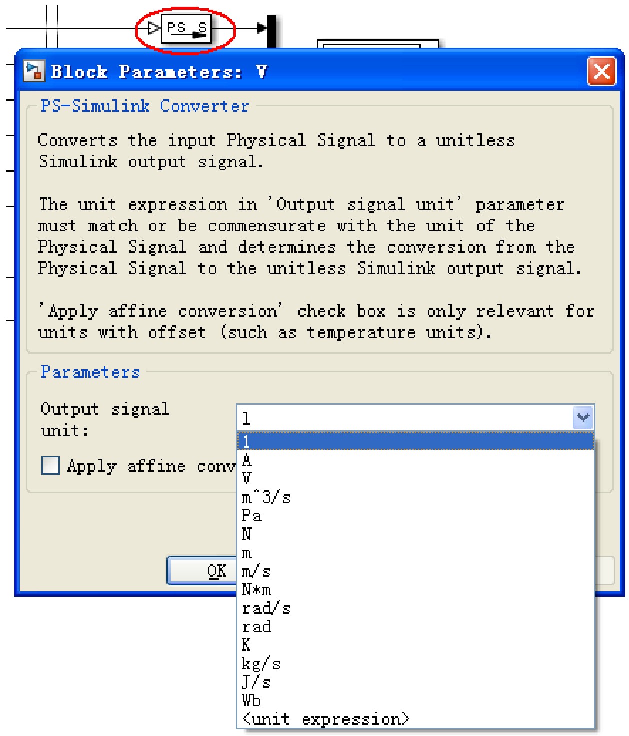


Figure F-32 Output converter of the Heater

The input signal port "mode" is linked by a constant block, where 0 (normal operation) or 1 (test mode) is set. In test mode, the value of flow rate uses the input of the signal port "Gin_fortest," which is from "F1 input." Also, by switch blocks, the inputs of the ports "Tin" and "I " are from the Lookup Tables "T1 input" and "C2 input."

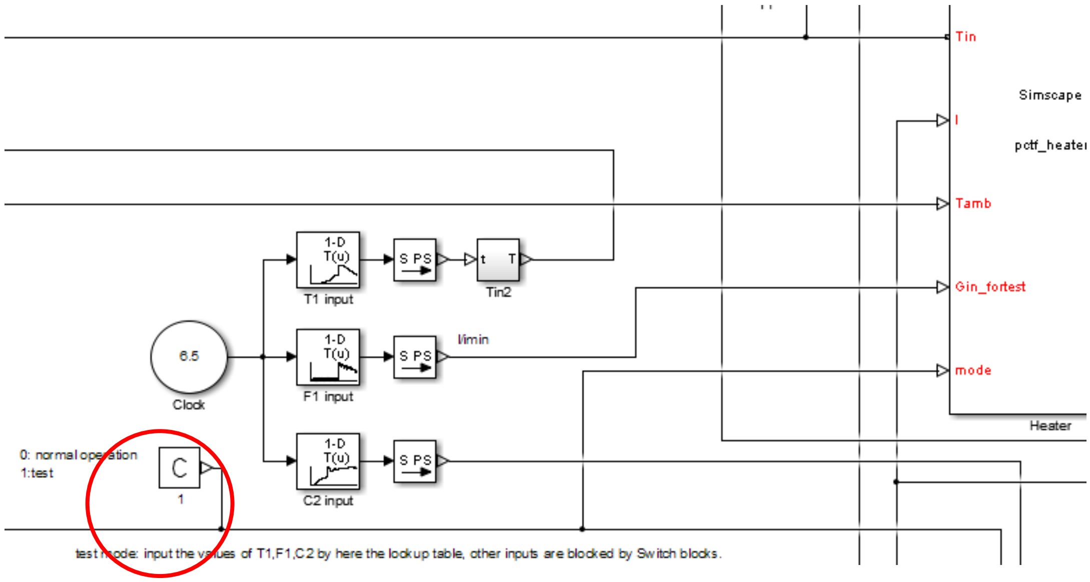


Figure F-33 Simulation mode of the Heater

2) Chiller

The Chiller subsystem is composed of the chiller block and related input and output blocks. Besides the lines connecting to other component blocks by conserving ports (symbol circle), there are several blocks to input and output variables by signal ports (symbol triangle).

In the Chiller system, the conserving ports “A” and “B” are connecting to the pipe and the valve in the primary water loop, while “A2” and “B2” are connected in the secondary water loop. The two connecting loops can form physical networks in the hydraulic domain. The pressure and flow rate are computed by solving the equation matrix of the physical network.

The conserving ports “Tin” and “Tout” are used to connect to the heater block through thermal inertia blocks, while “Tin2” and “Tout2” connect to the block of the Water Cooler. These ports work in the thermal domain, where the connections just transmit the values of the temperatures and there are no physical networks as the hydraulic domain and the pneumatic domain.

The output variable ports linked to the PS-Simulink Converter can convert physical signal to unitless signal. Commensurate values are converted in different units by selecting Output signal unit.

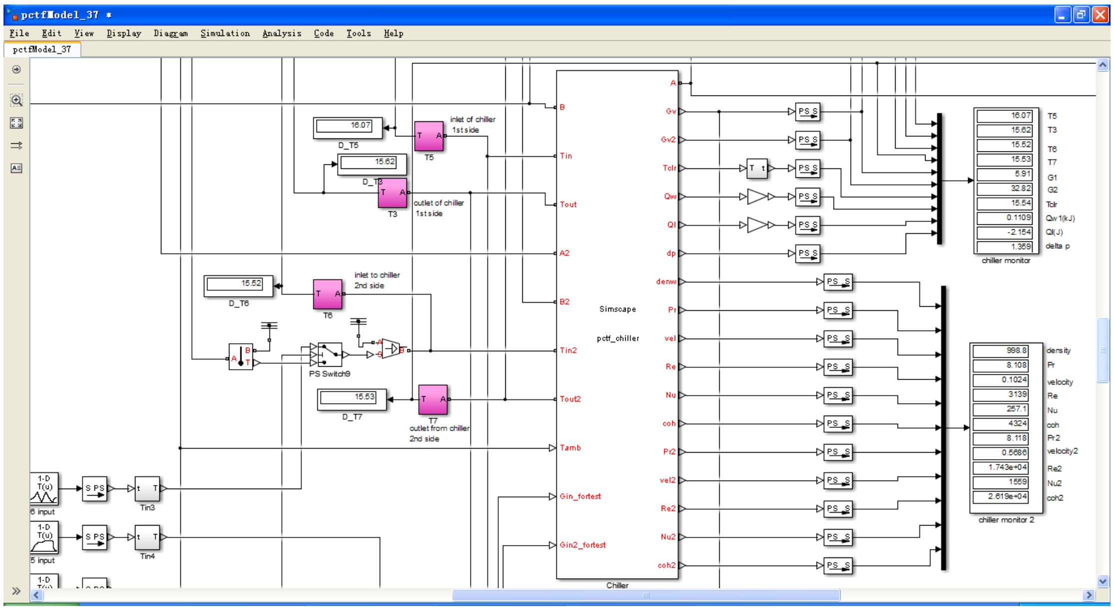
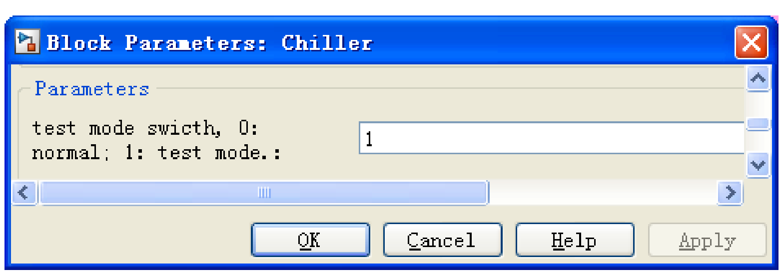


Figure F-34 Diagram of the Chiller

For test mode, two steps need to set: 1) test mode is set in parameters of the chiller block, where 0 means normal operation and 1 means test mode. In test mode, the values of flow rate use the input of the signal port "Gin_fortest" and "Gin2_fortest," which is from "F1 input" and "F3 input." 2) setting the constant block to 1 (test mode). This is to make the switch blocks use Lookup Tables "T5 input" and "T6 input" to input temperatures into ports "Tin" and "Tin2." To return to normal mode, the two setting need to be reset to 0. As the result, input temperatures and flow rates are then linked to normal work situation, and all the values from Lookup Tables are ignored.



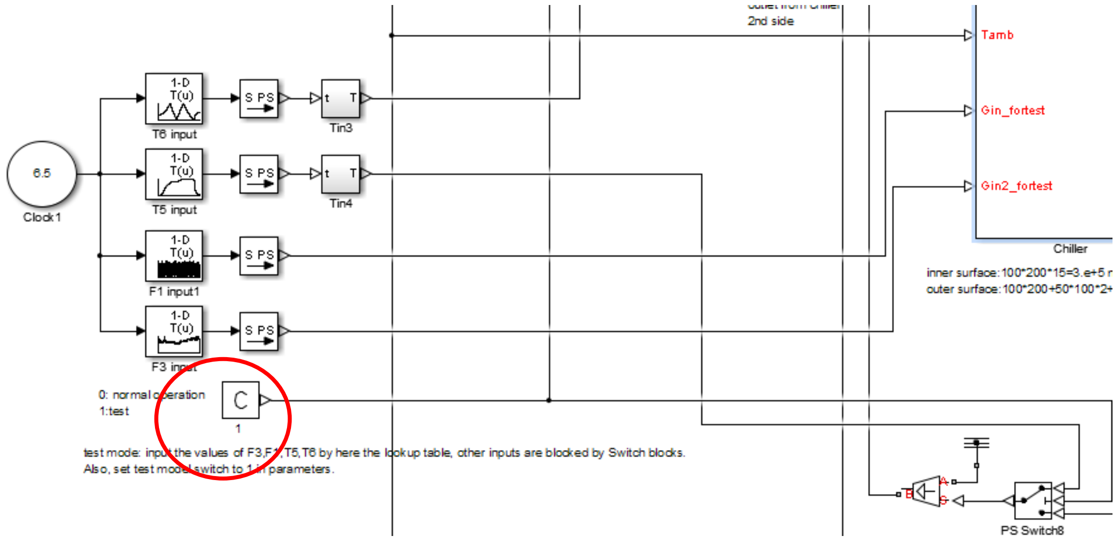


Figure F-35 Simulation mode of the Chiller

3) Thermal inertia

Two thermal inertia blocks between the thermal ports of the Heater and the Chiller are used to simulate the delay of thermal transmitting process. The delay is due to the thermal capacity of the water in the pipes.

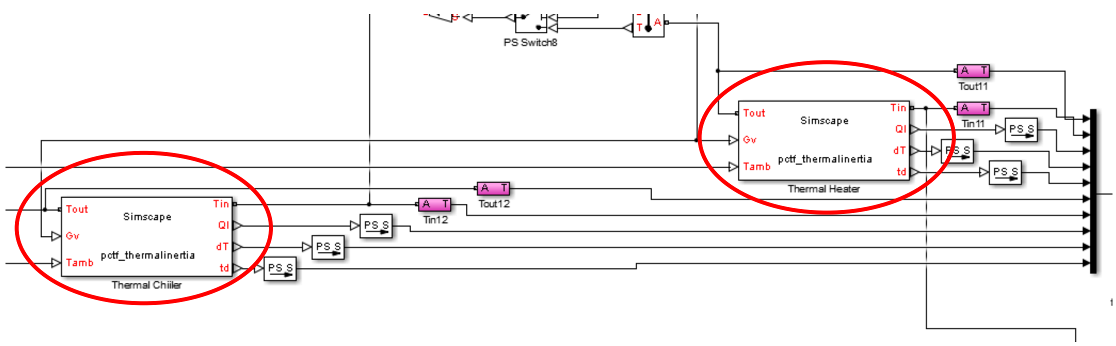


Figure F-36 Diagram of Thermal Inertia

4) Water Cooler

By employing a thermal inertia block, the process of heat diffused to the ambience in the Water Cooler is demonstrated. In the block, conserving thermal ports “Tin” and “Tout” connect to the Chiller, and the flow signal port “Gw” is connected to the flow meter F3 in the secondary water loop, and the temperature signal port "Tamb" is connected to the ambient temperature block.

Figure F-37 Diagram of the Water Cooler

In the parameter dialog box of the water cooler, the parameter "heat transfer coefficient of heat lost" needs to be given with a sufficiently large value, which is to ensure the capacity for heat diffusion.

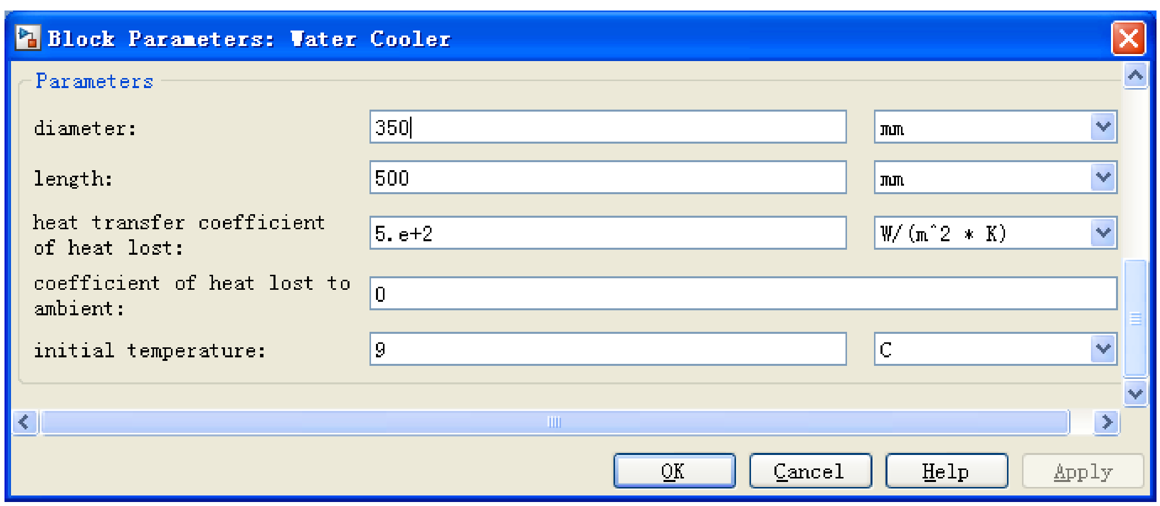


Figure F-38 Parameter of the Water Cooler

5) Pressurizer

The Pressurizer works in the hydraulic domain and the pneumatic domain as boundaries. Different from the Heater, the Chiller or the Turbine, the Pressurizer tank has no resistance. It contains the volumes of water and air, and has inner pressure.

The conserving pneumatic ports "Ain" and "Aout" are connected to the air input loop and the air output loop (spill loop). The conserving hydraulic port "Win" is connected to the valve CV-11, and through CV-11 linked to primary water loop. The conserving hydraulic port “Wout” is connected to the upper tank by the valve CV-12.

The pressures at the conserving ports are absolute pressures. The air measured pressures are gage pressures in which the atmospheric pressures are subtracted. Also, the output variables “pa” and “pw” of the Pressurizer block are gage pressures.

The output variable ports linked to the PS-Simulink Converter can convert physical signal to unitless signal. Commensurate values are converted in different units by selecting “Output signal unit.” The data types in the monitor are long real, which can show the very small changes of the air parameters.

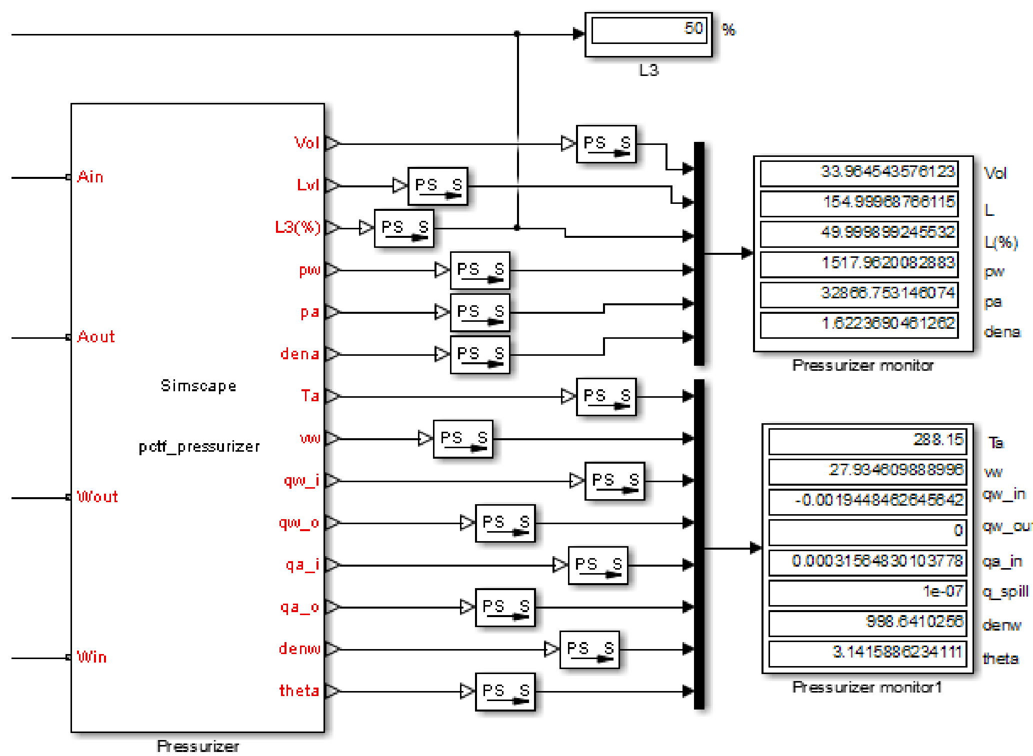


Figure F-39 Diagram of the Pressurizer

The initial air temperature and water level in the Pressurizer tank can be inputted in the parameter dialog box. Sometimes they are not the default ambient temperature (*20 ℃*) and zero water level. To ensure the simulation system is stable when executed from initial condition, the pressure of air is anticipated to start with the atmosphere pressure

(*101325 Pa*). To achieve this, the initial temperature of air is set as the same as that set in the air system. Then the initial mass of air is the correct value as the initial value of its differential equation.

6) HX-Tank

HX-Tank works in the hydraulic domain and the pneumatic domain as boundaries. Different from the Heater, Chiller or Turbine, the HX-Tank has no resistance. It contains the volumes of water and air, and it has varied inner pressures.

The conserving pneumatic ports “Ain” and “Aout” are connected to the air input loop and the air output loop (spill loop). The conserving hydraulic ports “Win” and “Wout” are connected to the filling water loop and the draining water loop.

The pressures at the conserving ports are absolute pressures. The air measured pressures are gage pressures in which the atmospheric pressures are subtracted. Also, the output variables “pa” and “pw” of this block are gage pressures.

The output variable ports linked to the PS-Simulink Converter can convert physical signal to unitless signal. Commensurate values are converted in different units by selecting “Output signal unit.” The data types in the monitor are long real, which can show the very small changes of the air parameters.

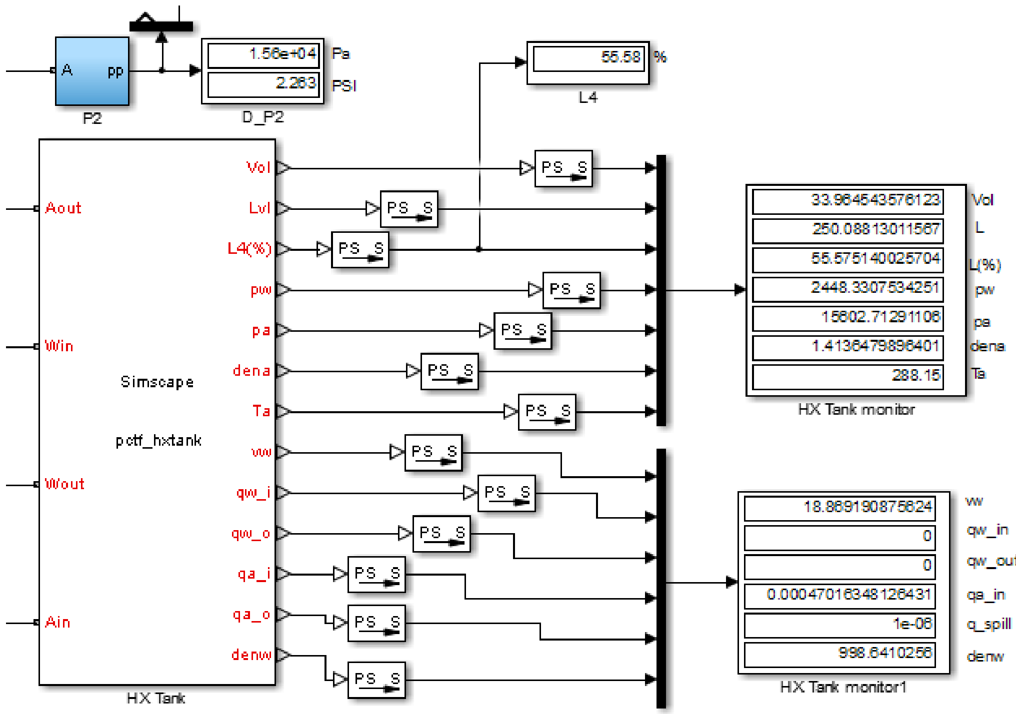


Figure F-40 Diagram of the HX-Tank

Different from the Pressurizer tank, the air inlet of HX-Tank is at the tank bottom. The air entered the upper space of the tank through the water, which produces air bubbles. The influence from the bubbles to the water level can be adjusted by the constant parameter “bubble coefficient.” This is referred with its mathematical expressions.

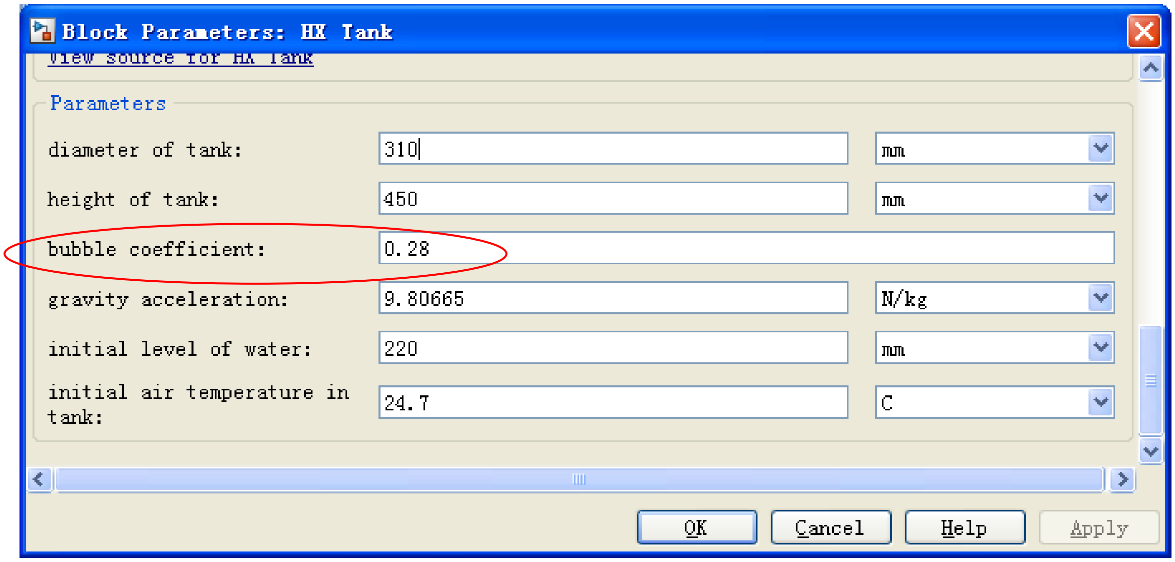


Figure F-41 Parameter of the HX-Tank

4.5.4 Water loops

As the below figure shown, the entire water loop can be considered to be composed of primary water loop, the filling water loop of the HX-Tank, the draining water loop of the HX-Tank, and the draining water loop of the Pressurizer. Each loop has hydraulic boundaries and hydraulic branches and is formed by connecting their conserving hydraulic ports.

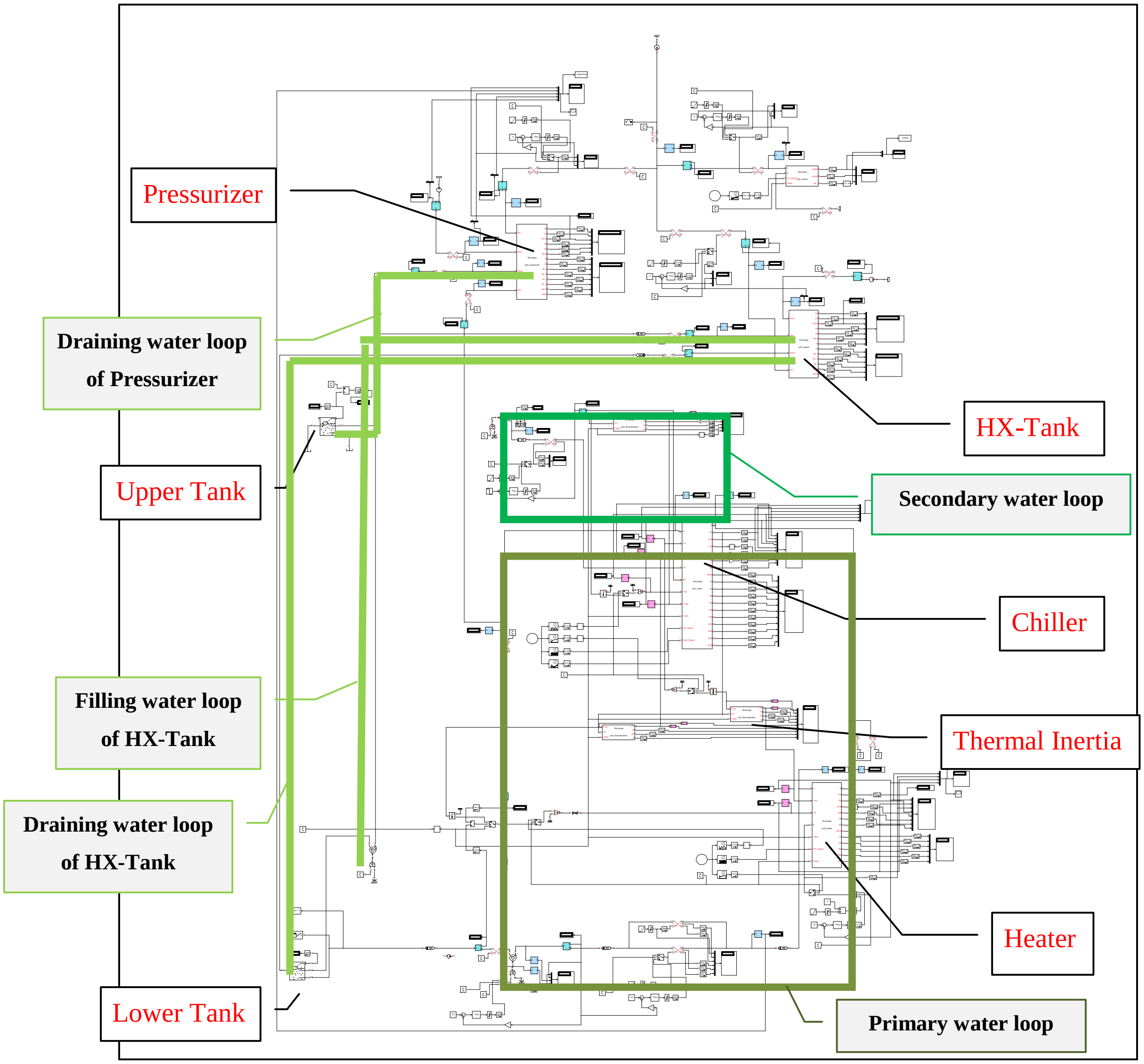


Figure F-42 Water loops

4.6 Diagram of the air loops

The pneumatic blocks used in the pressurized air loop are presented in this section. The guides of those I&C blocks and math blocks can be referenced in Simulink User Manual.

4.6.1 Sensor blocks

1) Flow meter of air

The flow meter of air is used to measure flow rate of air. The measured flow rate is positive if air flowing from port A to port B. The output port F includes two values with different unit of flow rate (*kg/s, l/Min*). This port can be linked to a Display block to show values. The output port Q is heat flow between the two pneumatic ports A and B, which is not used in the models.

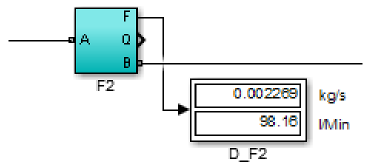


Figure F-43 Flow meter of air

The subsystem of this measurement can be opened by double-clicking the block. The block Pneumatic Mass & Heat Flow Sensor is from Simscape library. The default unit of flow rate of air is mass flow rate (*kg/s*), which is converted to volumetric flow rate (*l/Min*) based on the assumption of ambient temperature as 20 degrees centigrade. The two outputs with two units of flow rate are integrated to a Mux block and then output to signal port “F.”

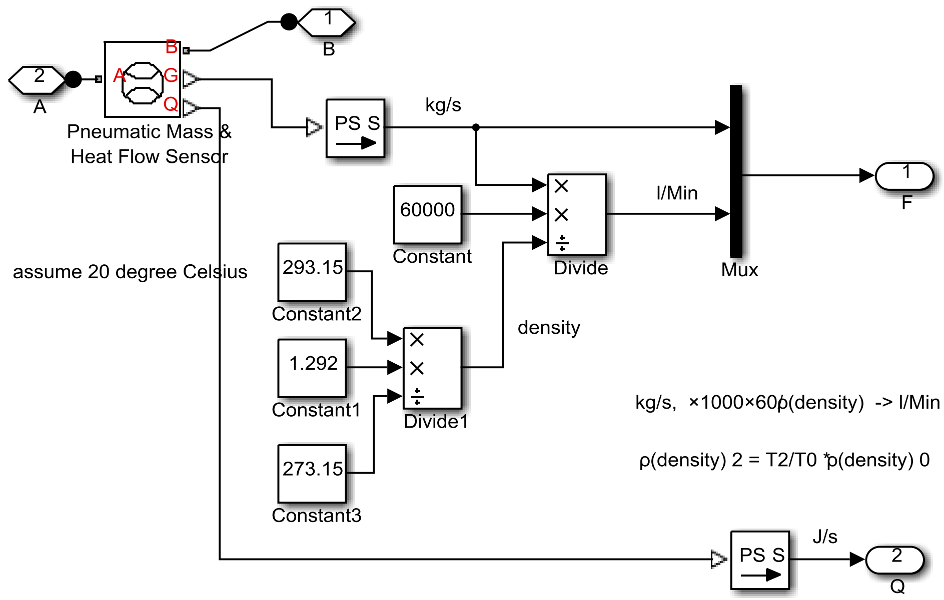


Figure F-44 Output of pneumatic mass sensor

2) Pressure sensor of air

The pressure sensor of air is used to measure gage pressure of air by attaching port A to the measuring point. The output port *pp* includes two values with two units (*Pa*, *PSI*). It can be linked to a Display block for displaying the values in running process.

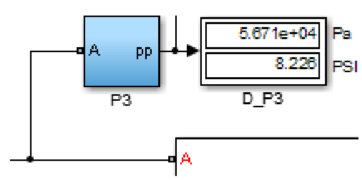


Figure F-45 Pressure sensor of air

The subsystem can be opened by double-clicking the block. The blocks of Pneumatic Pressure & Temperature Sensor and Pneumatic atmospheric Reference are both from Simscape library. According to the definition of pneumatic domain in Simscape, the ambient pressure is defined as *101325Pa*.

```
domain pneumatic
  parameters
    gam = { 1.4, '1' };                % Ratio of specific heats
    c_p = { 1005 , 'J/kg/K' };           % Specific heat at constant
pressure
    c_v = { 717.86 , 'J/kg/K' };       % Specific heat at constant
volume
    R   = { 287.05, 'J/kg/K' };        % Specific gas constant
    viscosity = { 18.21e-6, 'Pa*s' }; % Dynamic viscosity
    kin_viscosity = { 15.11e-6, 'm^2/s'}; % Kinematic viscosity
    Pa  = { 101325, 'Pa' };            % Ambient pressure
    Ta  = { 293.15, 'K' };             % Ambient temperature
  end
```

This can referenced with the block of Pneumatic Atmospheric Reference. Thus, the output pressures at port P are gage pressures. Chosen in two PS-Simulink blocks, the units of the output P are then separated into two types (*Pa*, *PSI*). Then, the two outputs with two units of pressure are integrated to a Mux block and output to signal port pp.

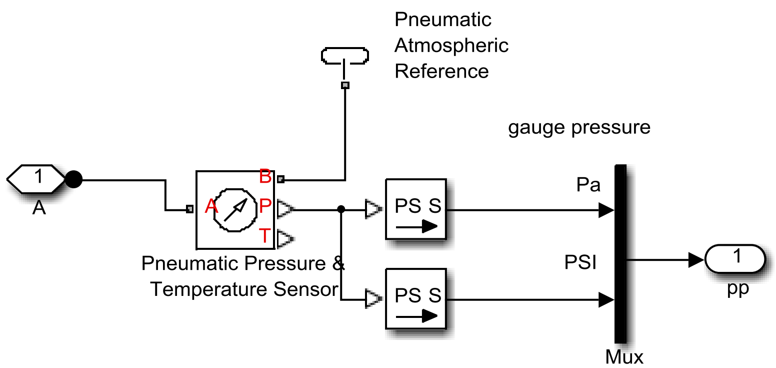


Figure F-46 Output of pneumatic pressure sensor

4.6.3 Main component blocks from custom block library

1) Turbine

Turbine is built as a pneumatic branch with some resistance. The output of air is linked finally to the boundary muffler which employs the block of Pneumatic Atmospheric Reference. The pneumatic resistance of muffler is added to the previous valve. Besides the conserving pneumatic ports (symbol circle) being used to connected air valves, there are two output signal ports (symbol triangle) to export the speed and the power.

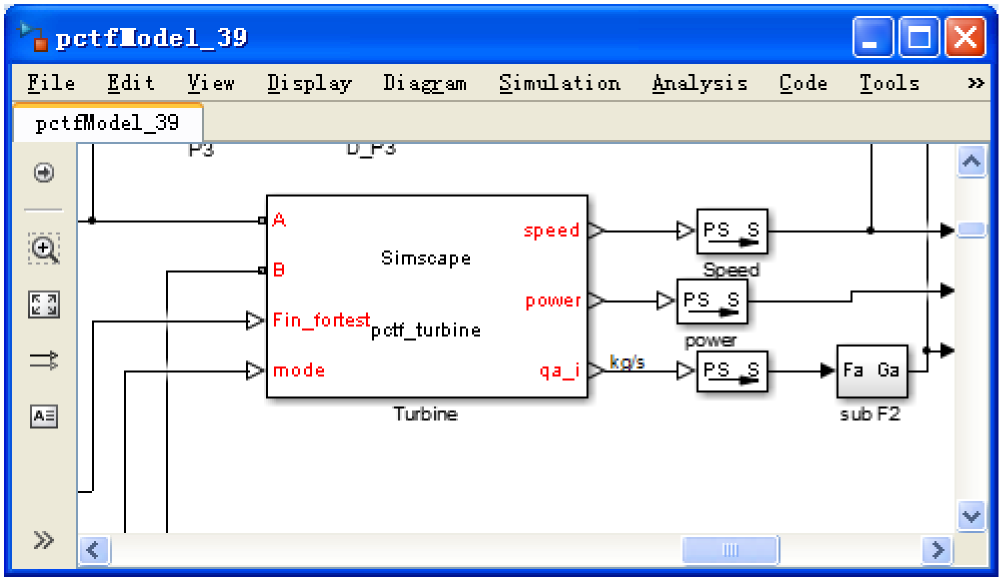


Figure F-47 Diagram of the Turbine

The output variable ports need to link to the PS-Simulink Converter to convert physical signal to unitless signal. Commensurate values are converted in different units by selecting "Output signal unit." The speed and the power use the units "*rpm*" and "*W*."

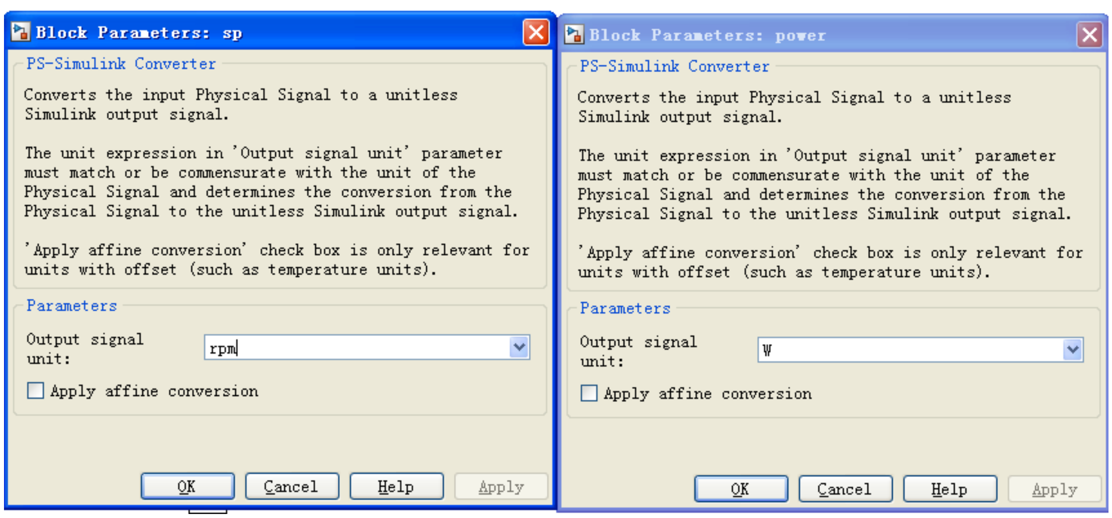


Figure F-48 Output of the Turbine

The input signal port "mode" is linked with a constant block, where 0 (normal operation) or 1 (test mode) can be set. In test mode, the flow rate is from the input of the signal port "Fin_fortest." This signal port is linked to "F2 input" through the block "Ga-Fa

Subsystem," in which the unit of flow rate is converted from *l/Min* to *kg/s*, and 20 degrees centigrade is assumed as the ambient temperature.

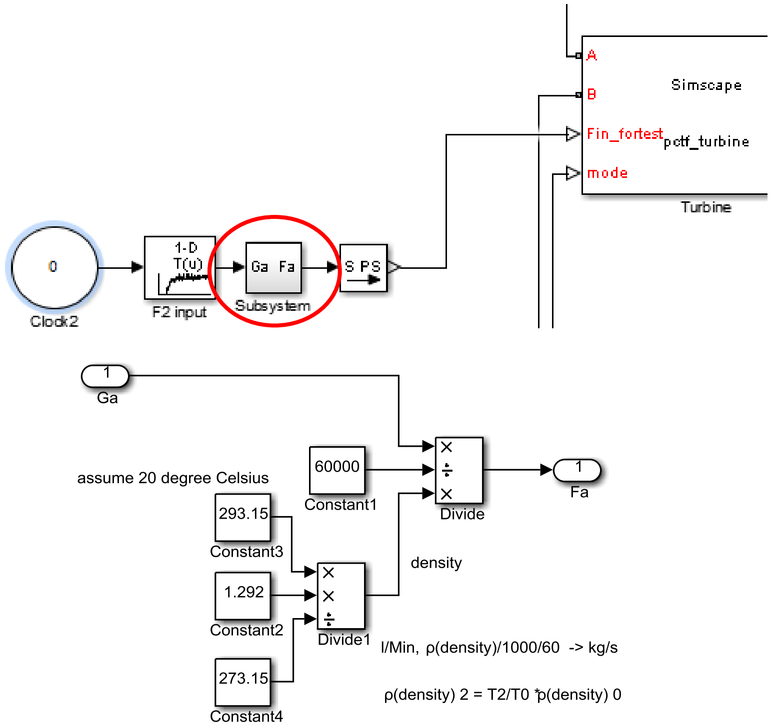


Figure F-49 Flow rate of air conversion

2) Pressurizer

As it is described in the section Diagram of water loops, the Pressurizer is also set as the boundary of the air loop. At its inlet and outlet lines, flow meters and pressure sensors are added for monitoring.

The port "Aout" is considered as the outlet for spilling. It is connected finally to a block of Pneumatic Atmospheric Reference to form an air loop.

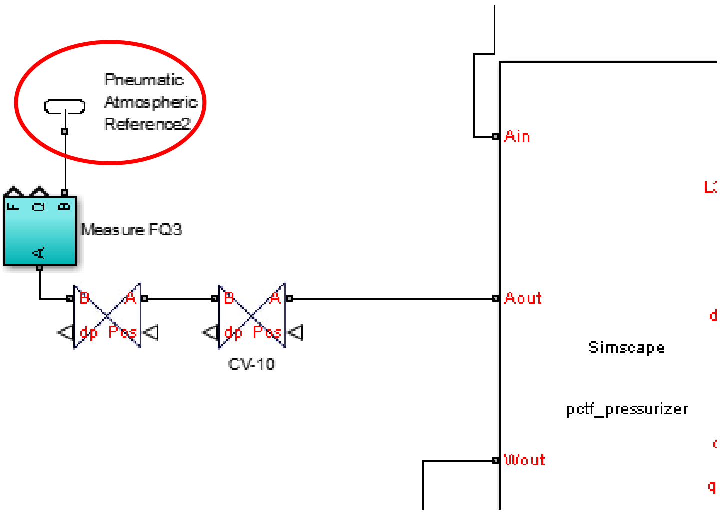


Figure F-50 Diagram of the air outlet of the Pressurizer

3) HX-Tank

The HX-Tank is introduced in the section Diagram of water loop. Additionally, the port “Aout” is considered as the outlet for spilling. This port is connected finally to a block of Pneumatic Atmospheric Reference to form an air loop.

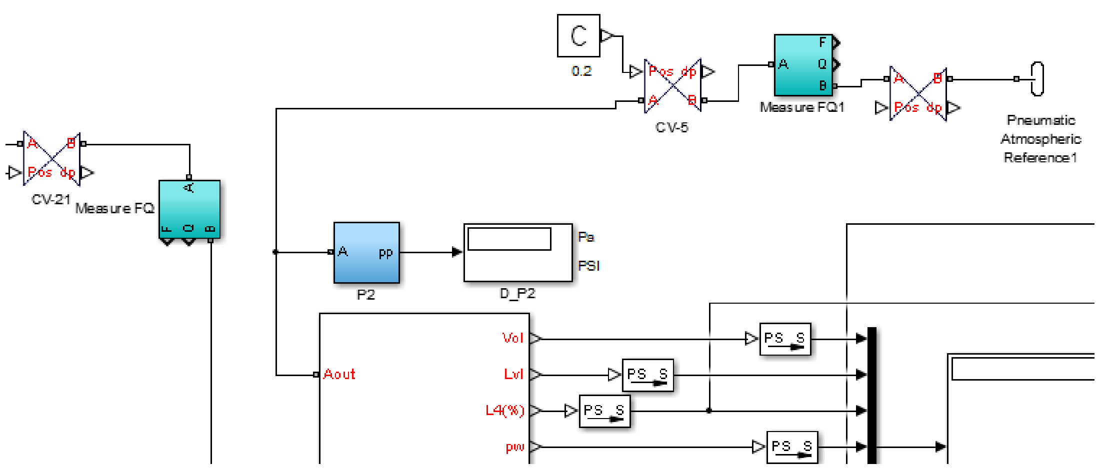


Figure F-51 Diagram of the air outlet of the HX-Tank

4.6.3 Air loop

The entire air loop can be considered to be composed of the main pressurized air loop, the spilling air loop of the Pressurizer, and the spilling air loop of the HX-Tank. All these loops have pneumatic boundaries and pneumatic branches. They are connected through the conserving pneumatic ports.

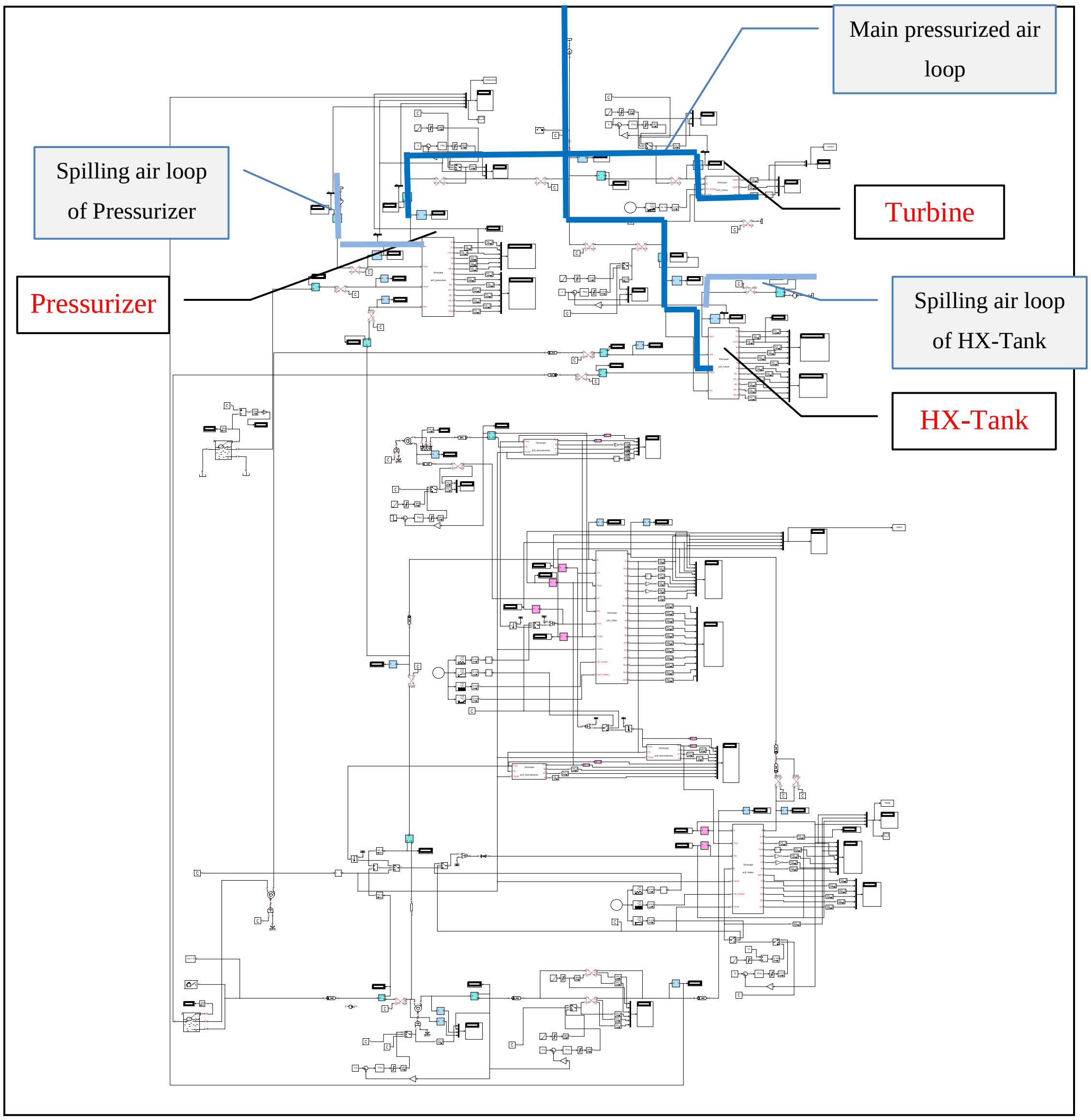


Figure F- 52 Air loop

The main pressurized air loop consists of a number of air valves and regulators. This loop starts from a block Pneumatic Atmospheric Reference with ambient pressure (*101325Pa*) and temperature (*293.15K*). Then, a pressure 40 *PSI* is added by a block Pneumatic Pressure Source from Simscape library. This is to simulate the pressurized air getting into the facility.

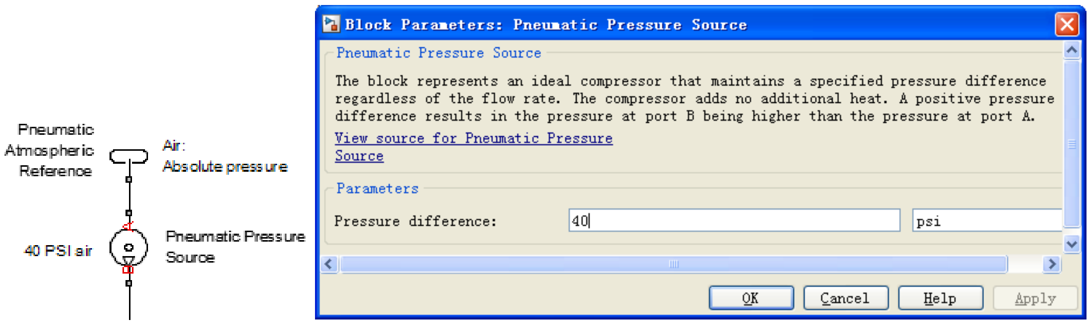


Figure F-53 Air source

To simulate the regulators, air valve blocks are employed with much larger pneumatic resistance.

4.7 Controllers

Three PID controllers are added in the diagram as the examples to test the controls in PID method, which are summarized in the following table. The other controllers in NPCTF can be added similarly as these existing control loops.

Description	Controlled Variable	Manipulated Variable
Heater Controller	T2	C2
Flow F1 Controller	F1	CV-1
Pressurizer Controller	P4	CV-9

4.8 Model configuration parameter

Before running the simulator system, the parameters of the model configuration need to be set. That clicking the menu opens the parameter window.

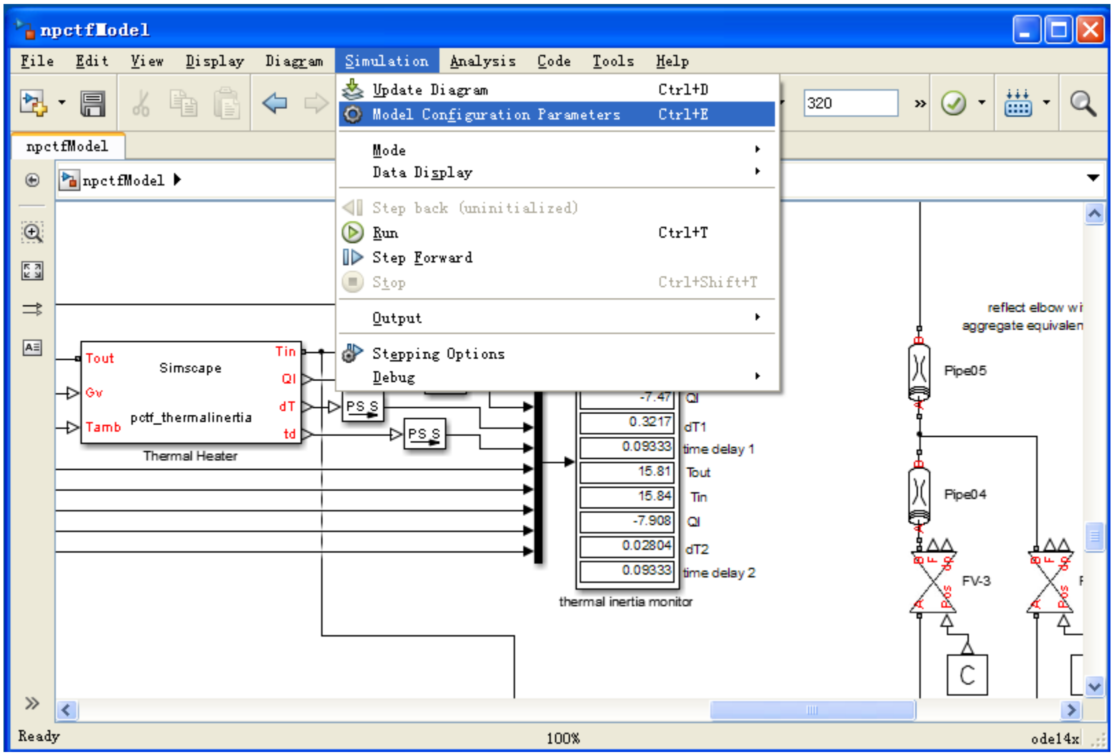


Figure F-54 Opening model configuration parameter

The simulation time is inputted in seconds. In the solver options, fixed-step type and ODE14X (extrapolation) is chosen. This solver may be the only solver of the ability to solve the model with abundant variables and equations. The fixed-step size (step time) can be set within a scope in one second. It can affect the iteration of differential equations. And of course the step time can influences the whole running time.

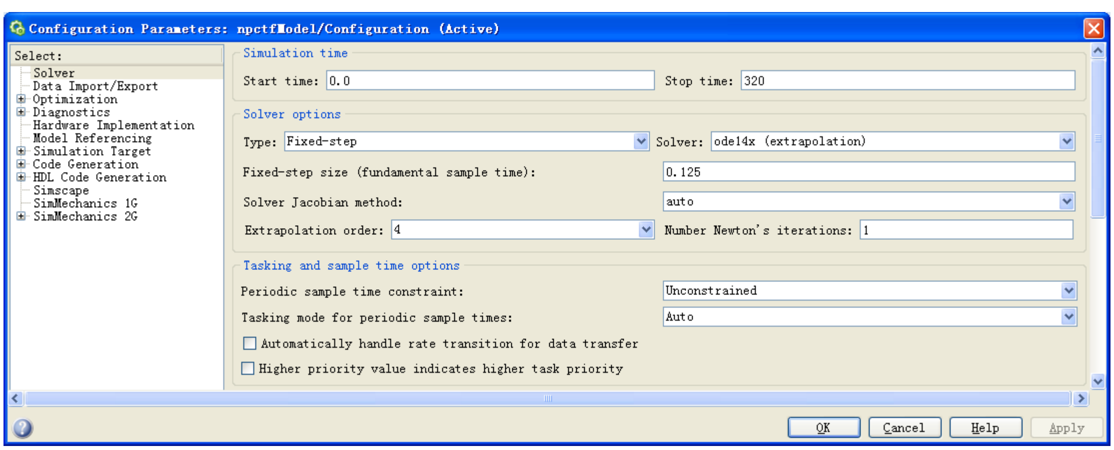


Figure F-55 Model configuration parameter

4.9 Unit conversions

Simscape manages the uses and conversions for physical units. In the component blocks, units are defined within the fields offered by Simscape. In the diagram of the simulator, the units can be converted by the block PS-Simulink Converter. The mathematical relationships of the unit conversions of pressure, flow rate, and temperature are explained as follows.

- Pressure sensors in the diagram use the default physical unit Pascal (Pa), while in NPCTF, the unit of pounds per square inch (PSI) is used. Their relationship can be expressed as:

$$[Pa] = 1.450377 \times 10^{-4}[PSI] \qquad (4\text{-}1)$$

- In the diagram of the hydraulic loop, flow meters use the default physical unit, cubic metres per second (m^3/s). Accordingly, the sampling data in NPCTF uses the unit, liters per minute (l/Min). Their relationship can be expressed as:

$$[l/Min] = [m^3/s] \times \frac{1000}{60} \qquad (4\text{-}2)$$

- In the diagram, the flow measurement in the pneumatic loops uses the mass flow rate (kg/s), while the volumetric flow rate (l/Min) is the measurement result in NPCTF. Their conversion can be depicted as:

$$[l/Min] = [kg/s] \times \frac{1000\times60}{\rho_a} \qquad (4\text{-}3)$$

When the ambient temperature is assumed as 20 ℃ (293.15 K), the air density ρ_a is determined as:

$$\rho_a = \frac{T}{T_0}\rho_0 = \frac{293.15}{273.15} \times 1.293 = 1.387(kg/m^3) \qquad (4\text{-}4)$$

where $\rho_0 = 1.293kg/m^2$ is the air density under the standard condition.

- For temperatures, the unit Celsius or Centigrade (℃) is converted from Kelvin (K) by:

$$[°C] = [K] - 273.15 \qquad (4\text{-}5)$$

- The custom-defined blocks include hydraulic resistances. The default unit is $Pa/(m^3/s)$. It can be converted to $PSI/(l/Min)$ by:

$$[Pa/(m^3/s)] = 2.4173 \times 10^{-9}[PSI/(l/Min)] \qquad (4\text{-}6)$$

As an example, if the hydraulic resistance is set at *7.5e+7*, the conversion is:

$$7.5e + 7\ (Pa/(m^3/s)) = 0.1813\ (PSI/(l/Min)) \qquad (4\text{-}7)$$

This means there exists a pressure drop of *0.1813 PSI* when *1 l/Min* water is flowing.

5 Running

5.1 How to run the simulator

Clicking the RUN button on the Toolbars or in the simulation menu to start running the model system. The simulation sequence can reference *Simscape User's Guide*. The running status can be seen on the Status Bar.

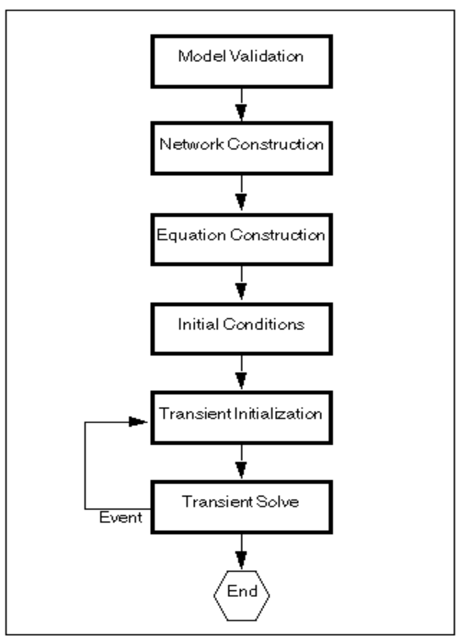


Figure F-56 Simulation sequence in Simscape

5.2 Initial conditions

During the running process, all the parameters cannot be changed. The initial conditions must be set before running. There are some essential initial parameters as described below.

5.2.1 Ambient temperature

Ambient temperature is mainly used to calculate the heat lost to the environment in the Heater, Chiller, Thermal inertia, and Water Cooler. These blocks all contain the signal port "Tamb" to be connected to the ambient condition. The ambient condition is a PS Constant block to input the value of temperature degree centigrade, and then the temperature is converted to degree Kelvin.

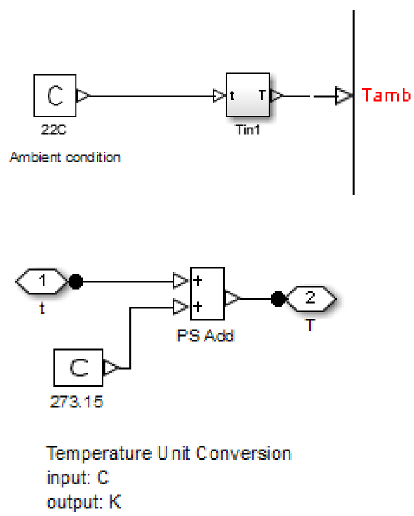


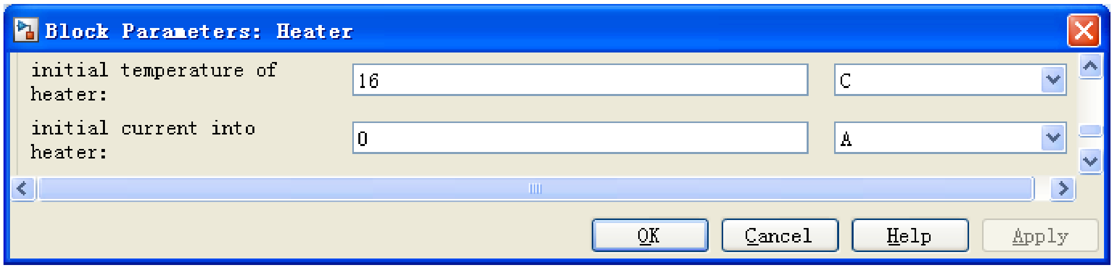
Figure F-57 Ambient temperature

5.2.2 Test mode or normal mode

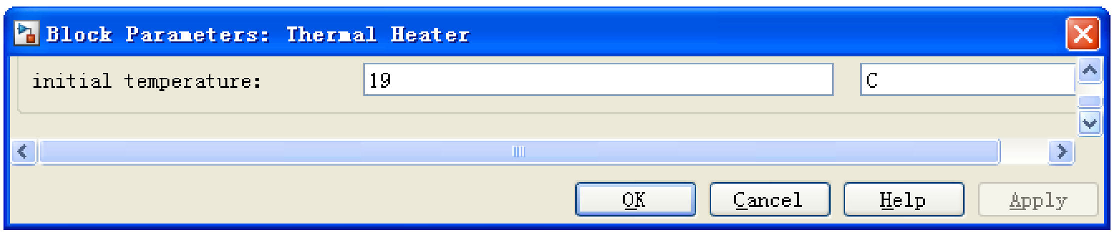
Test mode is used in the Heater, Chiller, and Turbine. If conducting tests on these components, "Test mode" should be chosen. Otherwise, all the modes should be set to "Normal mode," in which all the input variables are transmitted in the loops.

5.2.3 Initial parameters of components

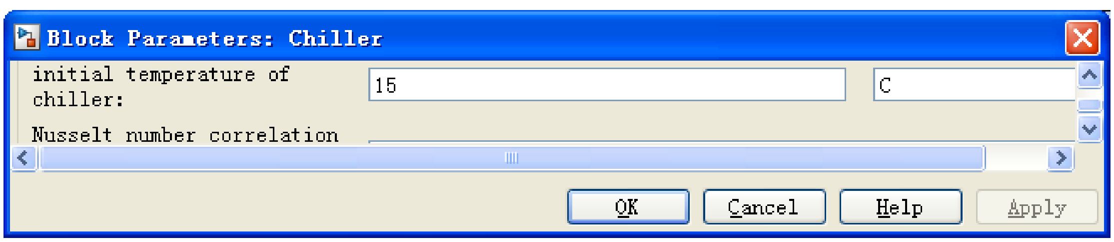
The initial parameters in blocks should be checked to ensure it stable in running the simulator system with correct initial conditions.



(a) Heater

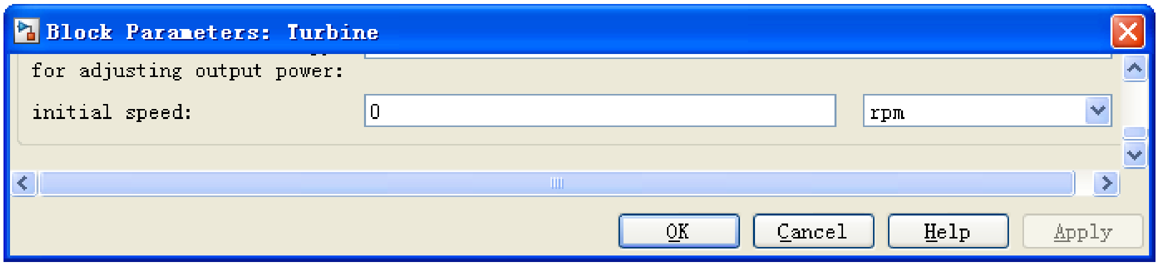


(b) Thermal inertia

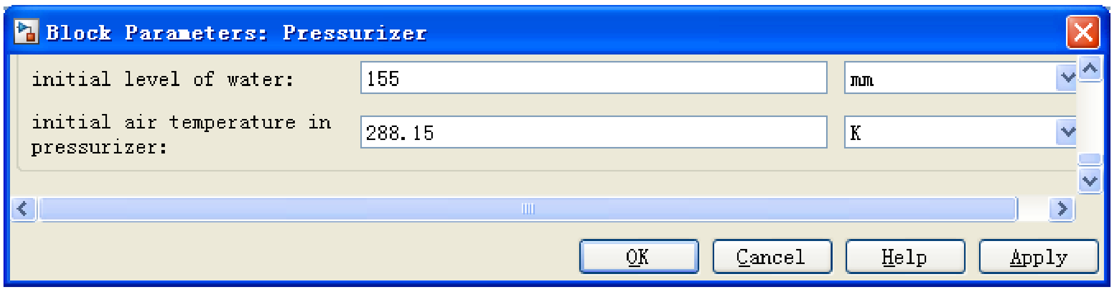


(c) Chiller

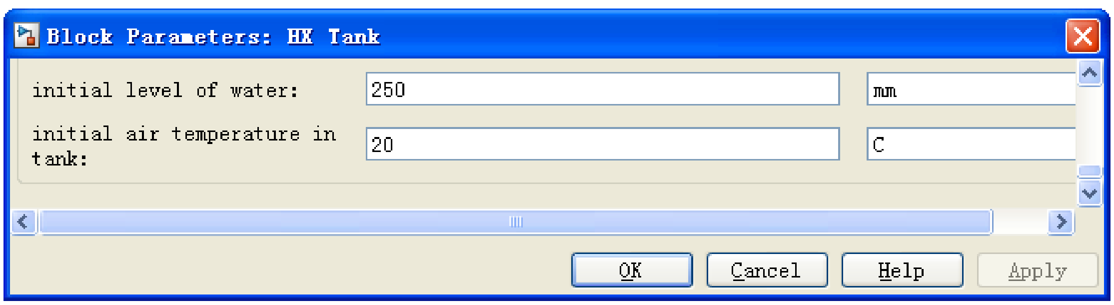
The initial temperatures of the Heater, Chiller, and Thermal inertia, as internal temperatures, are used as the initial values for the iteration of the Ordinary Differential Equations (ODEs) of metal temperatures and outlet temperatures. These initial values can make the iterative processes more quickly to converge on their lines.



(d) Turbine



(e) Pressurizer



(f) HX-Tank

Figure F-58 Initial condition settings

The initial water levels of the Pressurizer and the HX-Tank give the beginning level when the models start to execute. The initial air temperatures are used to compute the initial air mass in the tanks. These initial values aid to make the initial running process of the models more smoothly.

Reference

[1] Mathworks, *Simscape User's Guide R2014a*

[2] Mathworks, *Simscape Language Guide R2014a*

Curriculum Vitae

Name:	Binggang Cui
Post-secondary Education and Degrees:	Zhejiang University Hangzhou, Zhejiang Province, China 1993-1997 B. E. The University of Western Ontario London, Ontario, Canada 2013-2015 M. E. Sc.
Related Work Experience	Research Assistant The University of Western Ontario London, Ontario, Canada 2013-2015 Simulation Development Engineer Beijing Tongfang and Beijing Neoswise, China 2002-2011

Publication:

Binggang Cui and Jin Jiang, "Development of Dynamic Models for Thermal-hydraulic Processes based on Simscape Toolbox," 9th *International Topical Meeting on Nuclear Plant Instrumentation, Control, and Human-Machine Interface Technologies (NPIC & HMIT 2015)*, Charlotte, N.C., USA, Feb. 23-26, 2015.