Royal Institute of Technology

TRITA-FYS 2012:49

ISSN 0280-316X

ISRN KTH/FYS/--12:49—SE

**Validation of TRACE Code against
ROSA/LSTF Test for SBLOCA of Pressure
Vessel Upper-Head Small Break**

MIAN XING

Master of Science Thesis

Division of Nuclear Power Safety

Stockholm, Sweden 2012

Abstract

OECD/NEA ROSA/LSTF project tests are performed on the Large Scale Test Facility (LSTF). LSTF is a full-height, full-pressure and 1/48 volumetrically-scaled two-loop system which aims to simulate Japanese Tsuruga-2 Westinghouse-type 4-loop PWR. ROSA-V Test 6-1 simulates a pressure vessel (PV) upper-head small break loss-of-coolant accident (SBLOCA) with a break size equivalent to 1.9% of the volumetrically scaled cross-sectional area of the reference PWR cold leg.

The main objective of present thesis is to build a TRACE calculation model for simulating thermal hydraulic behaviors in LSTF and PV upper-head SBLOCA, so as to assess different modeling options and parameters of TRACE code. The results show that TRACE code well reproduce the complex physical phenomena involved in this type of SBLOCA scenarios. Almost all the events in the experiment are well predicted by the model based on TRACE code. In addition, the sensitivity of different models and parameters are investigated. For example, the code slightly overestimates the break mass flow from upper head which affects the accuracy of the results significantly. The rising of core exit temperature (CET) is significantly influenced by the flow area of leakage between downcomer and hot leg. Besides, the effect of the break location, low pressure injection system (LPIS) and accumulator setup are also studied.

Key words: SBLOCA, LSTF, TRACE, thermal-hydraulics transient, safety analysis

Acknowledgements

First of all, I would like to express my sincere gratitude to my supervisor Dr. Weimin Ma who guides my thesis work with his insight, patience and encouragement. I would also like to express my gratitude to Prof. Sevostian Bechta for his kind support and being my examiner.

I also want to express my gratitude to Liangxing Li and Shengjie Gong who give me a lot of comments about thesis writing in addition to the physical knowledge, experimental setup and pingpang technique, and Hua Li who teach me the convergence between SNAP and input deck, Sebastian who guide me familiar with the project and give me his model as template, and Sachin who shares his high speed computer, all this help me a lot during my thesis work.

Besides, I really appreciate the lab experience supported by Weimin Ma, Aram, Alexander and Storm.

I am very grateful to the whole department of Nuclear Power Safety who create such a harmony and hardworking circumstance in addition to all friendly and helpful staff here.

I want to show my thanks to all my friends in Stockholm: Chen Sisi, Zhang Youpeng, Chang Zhongwen, Shao Yiqiong, Zhong Huaqiang, Xu Zhenxiang, Li Gongzhuo, Sun Jing, Li Yang, Shi Chen, Zhu Yafeng, Liu Peng, Peng Lin and Zhang Zhongyuan, Paopao, Dingding etc.

Finally, I want to thank my parents who encourage and support my study here and my boyfriend, Hu Xiao who study and live with me every day, console and help me whenever I have difficulties.

Contents

1. Introduction.....	6
1.1 Safety analysis.....	6
1.2 LOCA	6
2. ROSA test 6-1	8
2.1 Background	8
2.2 ROSA facility-LSTF	8
2.3 ROSA-V project test 6-1	10
3. Simulation tool	16
3.1 TRACE code	16
3.2 Characteristics of TRACE	16
3.3 TRACE model theory	18
3.3.1 Component model.....	18
3.3.2 Control procedure	20
3.3.3 Critical flow model.....	20
3.3.4 Junction loss coefficient.....	21
3.4 TRACE range.....	23
4 Model simulation.....	24
4.1 Pressure vessel and its internals.....	25
4.2 Break unit	29
4.3 Pressurizer	29
4.4 Steam generators (SG).....	30
4.4.1 SG primary side.....	30
4.4.2 SG secondary side.....	30
4.4.3 SG relief valves.....	31
4.5 Other components of primary system.....	31
4.6 Emergency core cooling system (ECCS)	31
4.6.1 Accumulator	31
4.6.2 Low pressure injection system (LPIS).....	33
5. Results and discussions	34
5.1 Model validation: components geometry comparison	34
5.2 Steady state results.....	35
5.3 Transient calculation.....	35
5.3.1 Chronological sequence.....	36
5.3.2 Break unit mass flow	36
5.3.3 System pressure.....	38
5.3.4 Steam generator relief valves mass flow	40
5.3.5 Core exit temperature (CET)	41
5.3.6 Maximum fuel rod surface temperature	43
5.3.7 Primary loop mass flow rate.....	44
5.3.8 Collapsed liquid level in the core and upper head	46
5.3.9 ECCS injection mass flow	48
5.4 Sensitivity study.....	50

5.4.1 Break unit simulation.....	50
5.4.2 Leakage between downcomer and hot leg	53
5.4.3 Break location	55
5.4.4 Accumulator simulation	57
6. Conclusions.....	62
References.....	64

1. Introduction

1.1 Safety analysis

Nuclear power which uses nuclear fuel to generate heat and electricity is considered as a sustainable energy generation way. However, it also poses many threats to people and the environment and makes the nuclear safety analysis so significant. Nuclear safety analysis is the study of the reactor behaviors under the fault conditions. The events for safety analysis usually consist of the design basis accidents and beyond the design basis accidents [1]. Within the design basis accidents, required safety systems still work as intended under the malfunctions of the normal operating and control systems. However, during the events of beyond design basis, essential safety systems cannot be available as intended.

1.2 LOCA

Loss-of-coolant accident (LOCA) is one of the most important design basis accidents. The types of LOCA are characterized by different break flow area. Usually, a large LOCA has the flow area whose diameter is larger than 250 mm, a medium LOCA is between 80 and 250 mm, and small LOCA has a diameter of 10 - 80 mm [1]. In order to compensate the lost coolant, one or more emergency core cooling systems (ECCS) would be initiated, i.e. high-pressure safety injection, low-pressure safety injection and accumulators. High- and low-pressure safety injection systems and accumulator are actuated by different reactor pressure set points. For example, the accumulator will inject water to system as soon as the primary pressure drops to about 4.5 MPa.

Before the Three-Mile Island accident, the large-break LOCA was considered as the design basis accident for PWR, which attracted most attention of the safety analysis of postulated accidents. However, the occurrence of TMI-2 accident reminded people the danger of small-break LOCA and led to more and more detailed analysis of the small-break LOCA [2]. During a small-break LOCA, the reactor depressurization is much slower than the large-break LOCA which makes it impossible for LPIS to inject large rate of coolant flow at beginning. Besides, it is more likely to occur so it is worthy to study small-break LOCA (SBLOCA).

Typically, the sequence of events happened in SBLOCA proceeds as follows: control rods will be inserted into the core completely and the scram occurs within 20 to 60 seconds as soon as the reactor pressure drop is sensed. Then, the main coolant pump will be stopped and feed water will be also switched off. Although the fission power is shut down, the decay heat is still released. To take it away, the auxiliary feed water system is taken into operation. When the pressure drops to 10 MPa, the HPIS will pump water with a small flow rate to the core and it is the water phase in the core. After that, when the pressure falls below saturation pressure, the water inside the core turns to be saturated and it is the two-phase mixture of steam and water in the core. About 10 - 15 minutes later, the pressure drops to about 4 - 4.5 MPa and the

accumulator will be initiated. Eventually, the pressure would be stabilized at about 1 MPa and the low-pressure injection system could pump water into the primary system, so the core would be rapidly reflooded to be a cold condition [2]. The injection continues until the water in the storage tank is used up.

After the scram, the core is cooled by natural circulation. Firstly steam bubbles form and settle in PV. After the water level falls below the hot leg nozzle, the steam escapes to the steam generators (SG) and is condensed since the coolant temperature in SG secondary system is much lower. Then the condensed water will flow back to the reactor vessel in the opposite direction ('reflux condenser mode') [1]. Such a phenomenon happens in the condition that the secondary side has to be at low pressure, and at a correspondingly low saturation temperature. But when the primary pressure decreases below the secondary pressure, no condensation can take place [2].

During the SBLOCA, the depressurization is much slower than previous case and the pressure stabilizes at a higher value than the low-pressure safety injection initiation pressure. So the reactor operator should reduce the pressure and temperature to activate the LPIS by using steam generators, auxiliary feedwater and opening the steam relief valves. As an alternative, the operator could manually initiate the main feedwater systems in one loop and use the other heat sink, i.e. turbine condenser. Because in SBLOCA, the break flow is not sufficiently large to take out the decay heat, alternative heat sinks are required to discharge or damp the steam either on secondary side or to the turbine. Another method to reduce the reactor pressure is to open and close the pressure relief valves repeatedly in the pressurizer.

In this study, TRACE code is employed to estimate the thermal hydraulic responses after a SBLOCA of pressure vessel (PV) upper head under the assumption of total failure of high pressure injection system (HPIS). The computational calculation results will be compared with the experimental data: I) to interpret the transient thermal hydraulic performance of SBLOCA of PV upper head on LSTF; II) to assess the predictability of TRACE code against the complex phenomena during the transient; III) to conduct sensitivity study and find out the influence of some parameters or models.

2. ROSA test 6-1

2.1 Background

The ROSA program was initiated in 70s of last century for the study of thermal-hydraulic phenomena of light water reactors (LWRs) during loss-of-coolant accidents (both design basis LOCAs and small-break LOCAs) and operational transients, including fundamental blow down phenomena and effectiveness of emergency core cooling systems (ECCSs) by using scaled test facilities [3].

In 1991, ROSA project had progressed to ROSA-V program. The main objectives were defined to evaluate the effectiveness of Accident Management (AM) in the scenario of beyond design basis accidents (BDBAs), find out better solution to prevent severe accidents, and simulate passive safety systems simulation as well as the validate and improve computer codes [3].

2.2 ROSA facility-LSTF

During ROSA-V program, Large Scale Test Facility (LSTF) is the main test facility [3-4]. It was designed to model Japanese Tsuruga-2 3423 MWt (1160 MWe) Westinghouse-type 4-loop PWR by a full-height, full-pressure and 1/48 volumetrically-scaled two equal-volume-loop system with a core of 10 MW electrical heater. Hundreds of tests had been performed on LSTF to investigate various PWR plant behaviors and the effects of plant recovery methods for a wide spectrum of accident and transient conditions. In ROSA-V, LSTF is remodeled to have new fuel assembly design, full-height pressurizer and improved ECCS features which aims to confirm the effectiveness of various accident management (AM) measures for severe core damage prevention [3]. The effectiveness of passive safety systems could also be simulated and analyzed by using the device.

The main design features of remodeled LSTF and referred PWR are listed in Table 2.1. The core height, operation pressure and temperature are preserved. The geometry of single fuel rod is also in the same size, including rod diameter and the rod pitch. The power ratio between LSTF and referred PWR is 1/342. It is realized by different rod bundle size and different number of rods. For the reference PWR, the rod bundle size is 7×7 , while the LSTF is 17×17 . Each loop contains an active 1/24 volumetrically scaled steam generator (SG) and primary coolant recirculation system, since LSTF is designed to use 2-loop to simulate the 4-loop PWR with the 1/48 volumetrical scale. The length of SG tube is kept while the U-tube number is cut down to be 1/24 of the referred one. Besides, in order to better simulate the two-phase flow regime in the whole systems, the loop horizontal flow areas (i.e. the flow area of hot leg) are also sized to conserve the ratio between the length and square root of the diameter as following equation:

$$\frac{L_{PWR}}{\sqrt[2]{D_{PWR}}} = \frac{L_{LSTF}}{\sqrt[2]{D_{LSTF}}} \quad (2-1)$$

Figure 2.1 shows a general layout of the LSTF which consists of primary and secondary coolant system, including pressure vessel (PV), primary loop A with pressurizer (PZR) and symmetrical loop B without PZR, two steam generators, jet condenser, feedwater pump, auxiliary feedwater pumps and associated piping system. Besides, emergency core cooling systems (ECCSs) are made up by higher pressure injection system (HPIS), two accumulators (ACC-hot and ACC-cold) and low pressure injection system (LPIS).

Table 2.1 Main design features of LSTF and referred PWR.

Items	PWR	LSTF	PWR/LSTF
Core height (m)	3.66	3.66	1
Pressure (MPa)	16	16	1
Temperature (K)	598	598	1
Number of fuel rods	50952	1008	50.55
Rod bundle size	7×7(Square)	17×17	
Rod diameter (mm)			
Heater rod	9.5	9.5	1
Non-heating rod	12.24	12.24	1
Rod pitch (mm)	12.6	12.6	1
Total core power	3423(t)	10	342
Hot leg volume ($\frac{\pi}{4}D^2L$)	2.98	0.124	24
Hot leg $L/\sqrt{D}$	8.14	8.11	1
Number of primary loops	4	2	2
Number of SGs	4	2	2
Number of tubes in Steam Generator	3382	141	24
Length of Steam Generator tube (Average)(m)	20.2	20.2	1

Figure 2.1 General view of LSTF.

2.3 ROSA-V project test 6-1

ROSA-V project test 6-1 simulates a pressure vessel (PV) upper head small break loss-of-coolant accident (SBLOCA) with a break size equivalent to 1.9% cold leg break, as shown in Figure 2.2. It has been reported from Davis-Besse Reactor that the coolant and operating conditions in the primary system of PWR plant may cause the stress corrosion cracking of control rod drive mechanisms (CRDM) and instruments which are made from nickel-based alloys (i.e. Alloy 600) related weld metal [4]. The cracking process is called primary water stress corrosion cracking (PWSCC) which could result in the pressure vessel head wall thinning and degradation, and impose a threaten to the safety concerning vessel structural integrity [5]. Due to the circumferential cracking of the penetration nozzles for CRDM, there might be some leakage or a small break loss of coolant accident (SBLOCA) at the upper head of PWR. So a SBLOCA with a break size equivalent to 1.9% cold leg break at the PWR pressure vessel (PV) upper head is simulated in Test 6-1.

In the test, the break unit contains a horizontal pipe and a break valve, as shown in Figure 2.3. The inner diameter of the pipe is 87.3 mm and it locates at 435 mm higher than upper head

break nozzle. The break size is realized by the sharp-edge orifice fixed at the downstream of the pipe of 13.8 mm in diameter makes the orifice flow area correspond to 1.9% cross-sectional area of the cold leg in LSTF.

Figure 2.2 Location of SBLOCA at PWR PV upper head.

Figure 2.3 Experimental scheme of break units.

The test chronology is summarized in Table 2.2. During the experiment, the transient scenario begins with the opening of a valve on the top of upper head to simulate the SBLOCA. Because of the break, the primary pressure suffers a slow drop. When the lower plenum pressure reaches the set point of 12.97 MPa, the scram signal is triggered. In the referred PWR, when the reactor scrams, the control rods are inserted without delay. However, in the LSTF experiment, since there is no real fuel or control rod, the core power is adjusted manually as the postulated core power decay curve, as shown in Table 2.3 [6]. Besides, the primary pump is also coasted down, following a pre-determined rational speed curve. When the break initiates, the rotational speed almost doubles from 85.1 rad/s to 161.85 rad/s and stabilizes at this level. Until the scram signal, the rotational speed begins to slow down and comes to a stop at about 275 s. Once, there was controversy about whether the main recirculation pump should be stopped or not during LOCAs. The current rule is to stop the pumps. Not only because that they would promote the loss of fluid by circulating if they were left in operation, but also due to the fact that they would fail due to vibration. In addition to the power decay and pump coast down, when the reactor scrams, main feedwater of steam generators (SG) would be stopped, and turbine will be tripped by closing main steam isolation valves (MSIVs) in the steam line in secondary system. However, the closure of main steam isolation valves results in a rise of the pressure in the secondary system.

Shortly after scram signal, safety injection (SI) is initiated, when the PV lower plenum pressure drops to 12.27 MPa. As a result, auxiliary feedwater is initiated to pump subcooled water (300K) into the SG. After the moment, the steam relief valves (SRV) in both loops will switch on and off alternately to maintain the secondary pressure between 7.65 MPa and 7.95 MPa. At the same time, the pressure and temperature in the primary system will keep a relatively steady state because of the balance among the core power output, break loss and secondary side heat sink.

However, the balance will be broken as the upper plenum is depleted which leads to the phase change from two phase flow to single vapor in the break. There would be an immediate depressurization and a fast increase of temperature. When the core exit temperature (CET) is detected to reach 623K, the accident management (AM) will be initiated by fully opening of the SRVs in both SGs, which leads to a fast depressurization of the secondary system. But the AM action was ineffective in the early stage as long as the secondary pressure is higher than the primary pressure. As shown in Table 2.3, when the maximum fuel rod surface temperature or cladding temperature rises over 958K [4], specific core protection system logic would automatically reduce the core power to 10 % of the actual decay power.

In LSTF ROSA test 6-1, accumulators are designed as the main cooling measure. When the PV lower plenum pressure drop down to 4.5 MPa, the subcooled water (320K) from accumulator tanks condenses the vapor in the primary system which enhances the core reflooding and thus helps the core quench. After that, the PV lower plenum pressure decreases to 1.2 MPa, and the low pressure injection system is activated.

An accumulator consists of an accumulator tank and a stand pipe. LSTF has two accumulator tanks with the same size. The whole volume is 4.8 m³. There is 0.46 m³ initial nitrogen gas and

1.12 m³ water at the top of the tank. Below, there is the stand pipe of 4.2 m height. The accumulator check valve will be opened to inject water when the primary pressure drops to 4.51 MPa. Figure 2.4 illustrates the geometry and elevations of accumulator injection lines to the cold leg in both loops.

Table 2.2 Test chronology of the experiment.

Signal variable	Setpoint	Trigger	Event
Problem time	0	Break valve open	PV upper head break
PV lower plenum pressure	12.97 MPa	Generation of Scram signal	Initiation of core power decay curve
			Primary pump coastdown
			Main steam isolation valve closure
			Termination of main feedwater
PV lower plenum pressure	12.27 MPa	Generation of safety injection signal (SI)	Auxiliary feedwater start-up
Secondary side pressure	7.95 MPa 7.65 MPa	Safety relief valve open and close intermittently	Secondary system pressure oscillation between determined value
Core exit temperature	623 K	Accident management	Steam relief valve open permanently
Maximum fuel rod temperature	958 K	Core protection system activation	Core power is automatically decreased to 75%
PV lower plenum pressure	4.5 MPa	Accumulator valve open	Accumulator activates
PV lower plenum pressure	1.2 MPa	Low pressure injection system (LPIS) valve open	LPIS activates

Table 2.3 Predetermined core power decay curve.

Problem time (s)	Total reactor power (W)	Problem time(s)	Total reactor power (W)	Problem time(s)	Total reactor power (W)	Problem time(s)	Total reactor power (W)
0	10119000	20	10121000	46	10124000	47	9295000
48	7908000	49	7590000	50	7342000	70	4393000
90	3523000	200	2391000	600	1886000	1200	1465000
1201	1467000	1206	1348000	1207	1123000	1208	1103000
1209	1100000	1215	1096000	1216	846000	1217	270000
1218	189000	1219	170000	169900	162000	1300	159000
1400	159000	1900	144000	2290	129000	2291	108000
2292	100000	2293	96000	2671	98000	2706	119000
2707	123000	2708	134000	3165	137000	3265	137000

Figure 2.4 Accumulator injection line to cold leg in both loops.

It should be pointed out that some assumptions are put forward as the predetermined conditions as follows [4]:

- 1) The high pressure injection system fails and cannot inject any coolant into the system;
- 2) Loss of off-site power concurrent with the scram;

- 3) Considering the conservativeness of system, the threshold temperature of the core protection logic and controlling system is increased by 50K compared with realistic operation conditions. The scheme of core protection logic used in the test is shown in Table 2.4.

Table 2.4 Core protection logic.

Maximum fuel rod surface temperature (K)	Cut down the power to be
958K	75%
968K	50%
969K	25%
970K	10%
973K	0%

3. Simulation tool

Safety codes play a critical role in assessing the safety of a nuclear power reactor. Due to the sophistication of modern nuclear reactor systems, whose response to some proposed perturbation cannot be explained or predicted by human reasoning and simple theoretical model. Advanced computational tools are therefore required and developed to simulate the reactor behavior during operational transient and other accident scenarios. So far, several codes for LWRs have been developed for the safety analysis, such as RELAP5, RELAP5/PARCS, TRAC-AAA, EAC, SAS4/SASSYS, and SIMMER [7]. Also, a lot of comparisons of the simulation results have been performed using different computing codes [8 - 10]. TRACE is an advanced and best-estimate reactor system code for analyzing steady-state and transient thermal-hydraulic responses in LWRs [11 - 16]. In this thesis, TRACE version 5.0 patch 2 is implemented to analyze the ROSA SBLOCA tests.

3.1 TRACE code

TRACE (TRAC/RELAP Advanced Computational Engine) developed by the USNRC, consolidates the capabilities of four codes (TRAC-P, TRAC-B, RELAP 5 and RAMONA) into one. The code is capable of analyzing of LOCAs, operational transients, accident scenarios and phenomena in experimental facilities. TRACE uses different types of components which could be further subdivided into various numbers of physical volumes (or cells), to model each physical piece of equipment comprised in the reactor system. Over each finite volume cells, the fluid, conduction and kinetics equations are averaged. The hydraulic components in TRACE include BREAKs, CHANs (BWR fuel channels), VESSELs, PIPEs, FILLS, PRIZERs (pressurizers), HEATRs (feedwater heaters), PLENUMs, PUMPs, SEPDs (separators), TEEs, TURBs (turbines), JETs (jet pumps) and VALVEs and so on. Models supplied in TRACE include multidimensional two-phase flow, such as critical flow model, offtake model and counter current flow model, non-equilibrium thermo-dynamics, reflood, level tracking, reactor kinetics and generalized heat transfer [11]. The models and correlations in the codes play a key role in the speed of calculation and the accuracy of the results.

3.2 Characteristics of TRACE

According to the manual published by U.S.NRC [11] [17], TRACE mainly possesses the following characteristics.

Multi-Dimensional Fluid Dynamics

The component VESSEL is a 3D component set up in three dimensions Cartesian- or cylindrical coordinate, while PIPE and TEE are 1D component which can be used to model the flows within a coolant loop. The complex multi-dimensional flow networks can be simulated accurately by the combination of 1D and 3D components.

Non-homogeneous, Non-equilibrium Modeling

The six field conservation equations (gas-fluid) of hydrodynamic model are summarized as the following equation.

Time Averaged Mass Equation

$$\frac{\partial[(1-\alpha)\bar{\rho}_l]}{\partial t} + \nabla \left[(1-\alpha)\bar{\rho}_l \bar{\vec{V}}_l \right] = -\bar{\Gamma} \quad (3-1)$$

$$\frac{\partial(\alpha\bar{\rho}_g)}{\partial t} + \nabla \left[\alpha\bar{\rho}_g \bar{\vec{V}}_g \right] = \bar{\Gamma} \quad (3-2)$$

Time Average Energy Equation

$$\begin{aligned} & \frac{\partial \left[(1-\alpha)\bar{\rho}_l \left(\overline{e_l + V_l^2/2} \right) \right]}{\partial t} + \nabla \left[(1-\alpha)\bar{\rho}_l \left(\overline{e_l + V_l^2/2} \right) \bar{\vec{V}}_l \right] \\ &= -\nabla \left[(1-\alpha)\bar{\vec{q}}'_l \right] + \nabla \left[(1-\alpha) \left(\overline{T_l \vec{V}_l} \right) \right] + (1-\alpha)\bar{\rho}_l \bar{\vec{g}} \bar{\vec{V}}_l - \bar{E}_l + \bar{q}_{dl} \end{aligned} \quad (3-3)$$

$$\begin{aligned} & \frac{\partial \left[\alpha\bar{\rho}_g \left(\overline{e_g + V_g^2/2} \right) \right]}{\partial t} + \nabla \left[\alpha\bar{\rho}_g \left(\overline{e_g + V_g^2/2} \right) \bar{\vec{V}}_g \right] \\ &= -\nabla \left[\alpha\bar{\vec{q}}'_g \right] + \nabla \left[\alpha \left(\overline{T_g \vec{V}_g} \right) \right] + \alpha\bar{\rho}_g \bar{\vec{g}} \bar{\vec{V}}_g - \bar{E}_g + \bar{q}_{dg} \end{aligned} \quad (3-4)$$

Time Average Momentum Equations

$$\frac{\partial[(1-\alpha)\bar{\rho}_l \bar{\vec{V}}_l]}{\partial t} + \nabla[(1-\alpha)\bar{\rho}_l \bar{\vec{V}}_l \bar{\vec{V}}_l] = \nabla[(1-\alpha)\bar{T}_l] + (1-\alpha)\bar{\rho}_l \bar{\vec{g}} - \bar{M}_l \quad (3-5)$$

$$\frac{\partial[\alpha\bar{\rho}_g \bar{\vec{V}}_g]}{\partial t} + \nabla[\alpha\bar{\rho}_g \bar{\vec{V}}_g \bar{\vec{V}}_g] = \nabla[\alpha\bar{T}_g] + \alpha\bar{\rho}_g \bar{\vec{g}} + \bar{M}_g \quad (3-6)$$

In these equations, overbar represents time average, α is the probability that a point is occupied by gas phase, and Γ , E_i , and M_i represent the contribution of time averaged jump conditions to transfer of mass, energy and momentum respectively. Besides, q' is conductive heat flux, q_d is direct heating and T is the full stress tensor. The subscripts of "g" and "l" represent gas and liquid term respectively. A series of approximations are needed to complete the closure of these field equations.

Flow-Regime-Dependent Constitutive Equation Package

The above thermal-hydraulic equations describe the balance of mass, energy and momentum between two phase flows. In order to simulate and calculate different flow topology and

conditions which the two phase interactions depends a lot, a flow-regime-dependent constitutive equation package is incorporated into the code.

Comprehensive Heat Transfer Capability

Detailed heat transfer analyses of the components, i.e. vessel could be performed by TRACE. It means a 2D treatment of conduction heat transfer, quench fronts, fuel rod heat transfer, inner-surface and outer surface convection heat transfer even reactivity feedback could modeled.

Component and Functional Modularity

TRACE code uses the component-base approach. The hydraulic components in TRACE include BREAKs, CHANs (BWR fuel channels), VESSELs, PIPEs, FILLs, PRIZERs (pressurizers), HEATRs (feedwater heaters), PLENUMs, PUMPs, SEPDS (separators), TEEs, TURBs (turbines), JETs (jet pumps) and VALVEs and so on. All components are specified through input data. By the combination of the components, any LWR design or experimental configuration could be simulated. Besides, separate modules, such as heat transfer coefficient selection and basic hydrodynamics solution algorithm and other functions could be accessed and selected to perform different aspects of calculations.

3.3 TRACE model theory

3.3.1 Component model

TRACE uses a component-based approach to model reactor systems. Every physical piece of equipment could be represented as a component. All components used in the thesis are summarized in Table 3.1.

Table 3.1 TRACE model description.

Name	Description and application	Symbol
BREAK	To specify the pressure boundary conditions; other fluid state properties could be specified as constants or defined by signal variables or control blocks or trip.
FILL	To impose boundary conditions, such as coolant velocity, mass flow, fluid pressure, void fraction, temperature and so on, without modeling a real physical component of the system, i.e. feedwater flow.
VESSEL	To model a light water reactor vessel within associated internals, including the downcomer, fuel-assembly core, upper head, upper plenum and lower plenum. For vessel geometry, 1-,2- or 3D in Cartesian or cylindrical geometry could be selected. All fluid flow area and volume fraction and fluid conditions are input dimensionally specified.
HTSTR	To evaluate the dynamics of heat transfer, including conduction, convection and radiation or heat loss; Usually to model fuel rods inside a vessel or steam generator for which temperature distribution could be computed.
PIPE	Mostly to model general connectors between components or part of other structures, i.e. reactor vessel control rod guide tube (CRGT) or steam generator; also to accumulators by initiating some options.
VALVE	To model the flow through 1D pipe with predetermined changing area or to model various types of valves, such as check valve, trip controlled or controller activated valve.
PUMP	To describe and evaluate the interaction two-phase fluid with the simulation of centrifugal pump.
PRIZER	To simulate a PWR 1D pressurizer reservoir, including heater/sprayer logic to maintain the primary system pressure.
SEPD	To model BWR separator and the secondary system of PWR steam generator with TEE-like geometry by description of phase separation model.
POWER	To supply power to solid structures, for example, HTSTR component.

3.3.2 Control procedure

The TRACE control procedure is used to simulate the reactor control system and staff operation. It is flexible to specify a control procedure with desired control logic [17]. Table 3.2 summarizes the main control tools and its function. In the transient scenario, control systems are significant to accurately simulate the real situations.

Table 3.2 Description of main control tools.

Name	Description and application	Symbol
Signal Variables	Some parameters whose value are sent to its own output by a specialized control block	mode=exact value
Control Blocks	Functional operators to define the logic of a control system
Trips	ON/OFF switches to be used when: 1) to evaluate a component hardware action; 2) to define a status signal applied in a control block; 3) to define a coincidence trip or a blocking.	range type=1 signal variable s1=0.0, dt=0.0 (s) s2=0.0, dt=0.0 (s)

3.3.3 Critical flow model

The determination of choked flow rate is of significance in safety analysis of nuclear reactors [18], especially in the break when critical flow conditions are expected to appear. For single phase flow, the choked point where the flow is critical, is characterized theoretically by: 1) the flow rate has the maximum possible value; 2) the pressure of downstream from the choked point does not influence the flow or the fluid properties upstream from the choked point; 3) there are limited derivations of most of the fluid properties at the choked point.

When developing a TRACE model, choked-flow model could be activated by input the value '1' or '2' for choked-flow model option variable ICFLOW in NAMELIST data cards [19]. In addition, cell-edge choked-flow model option ICFLG also needs to be actuated for the specific component, i.e. valve simulating the break. In both ICFLOW and ICFLG, value of '1' means default multipliers for choked-flow model calculations. Value of '2' in ICFLOW means optional NAMELIST variable defined multipliers by users is turned on and in ICFLG, from value '2' to '5', four different groups of multipliers could be chosen by user to calculate the choked flow.

In TRACE, if $\alpha \leq 1.0 \times 10^{-8}$, the coolant would be considered as subcooled liquid and the cell-edge velocity is calculated as the maximum of the Bernoulli-predicted velocity for

homogeneous equilibrium sound speed [11]. If a sound-speed multiplier is specified in NAMELIST data by user, the final predicted liquid choking velocity is the product of maximum of the Bernoulli-predicted velocity with the multiplier.

For the cases of $1.0 \times 10^{-5} \leq \alpha \leq 0.999$, two-phase choked flow, an extension of model developed by Ransom and Trapp [20] will be used for iteration and calculation. The predicted value is calculated by the equation below.

$$V_{me}^p = \frac{\alpha_e \rho_{ge} V_{ge} + (1 - \alpha_e) \rho_{le} V_{le}}{\rho_{mo}} \quad (3-7)$$

$$V_{le}^p = CHMLT2 \cdot \frac{V_{me}^p \cdot \rho_{mc}}{\rho_{gc} \alpha_c S + \rho_{lc} (1 - \alpha_c)} \quad (3-8)$$

$$V_{ge}^p = S \cdot V_{le}^p \quad (3-9)$$

Where α_e is the cell-edge void fraction; ρ_{ge} and ρ_{le} are the cell-edge saturation densities determined by the cell-edge pressure; ρ_{gc} , ρ_{lc} , and ρ_{mc} are the cell-center steam/gas mixture, liquid and total mixture densities; CHMLT2 is a user-input chock-flow multiplier; S is the slip ratio which is calculated by the momentum-solution steam/gas mixture velocities and liquid velocities:

$$S = \frac{V_g}{V_l} \quad (3-10)$$

If the mixture velocity determined by the momentum solution is less than the predicted maximum liquid choking velocity, the flow will be treated as non-choked flow and choking calculation will be ended. However, as long as the momentum-solution velocity is greater than the maximum choking velocity, the steam/gas mixture velocity will be reset to the maximum choking values.

3.3.4 Junction loss coefficient

At entries and exits, orifices and area-change edges, energy irreversibly dissipates which is known as hydraulic resistance [8]. Although they are minor losses, they account for a major portion of total pressure loss. TRACE provides special models to specify the loss coefficients associated with them. As Bernoulli's equation shows the wall friction and local losses take up the pressure losses [21].

$$p_1 + \frac{1}{2} \rho v_1^2 + \rho g h_1 = p_2 + \frac{1}{2} \rho v_2^2 + \rho g h_2 + \frac{fL}{D_h} \frac{1}{2} \rho v_2^2 + \sum K \frac{1}{2} \rho v_2^2 \quad (3-11)$$

Where K is the coefficient of flow resistance, which is defined by:

$$\text{Flow Resistance} = \frac{\text{Total Pressure Loss}}{\text{Dynamic Pressure (Kinetic Energy)}} \quad (3-12)$$

So K-factor in TRACE simulation could be calculated as:

$$K = \frac{\Delta p_{total}}{\frac{1}{2} \rho v_{base}^2} \quad (3-13)$$

Where Base area refers to the smallest cross section and hence represents the largest velocity. Table 3.3 shows a list of configuration of different K-factor equation for main scenarios [21 - 23].

Table 3.3 List of different K-fact equation.

Scenarios	K-fact equation	Examples
Abrupt expansion	$K = \left(1 - \frac{A_1}{A_2}\right)^2$
Abrupt contraction	$K = 0.5 \left(1 - \frac{A_3}{A_1}\right)^{0.75}$
Sharp orifice	$K = \left[\left(1 - \frac{A_0}{A_1}\right) + 0.707 \left(1 - \frac{A_0}{A_1}\right)^{0.375} \right]^2 \left(\frac{A_0}{A_1}\right)^2$
Rounded orifice	$K = \left[\left(1 - \frac{A_0}{A_1}\right) + \sqrt{\zeta} \left(1 - \frac{A_0}{A_1}\right)^{0.75} \right]^2$ $\zeta = 0.03 + 0.47 \times 10^{-7.7 \frac{r}{D_{ho}}}$
Beveled orifice	$K = \left[\left(1 - \frac{A_0}{A_1}\right) + \sqrt{\zeta} \left(1 - \frac{A_0}{A_1}\right)^{0.375} \right]^2$ $\zeta = 0.03 + 0.47 \times 10^{- \left[3.4 \frac{1}{D_{ho}} + 88.4 \left(\frac{1}{D_{ho}} \right)^2 \right]}$
Thick orifice	$K = \left[0.5 \left(1 - \frac{A_0}{A_1}\right)^{0.75} + \tau \left(1 - \frac{A_0}{A_1}\right)^{1.375} + \left(1 - \frac{A_0}{A_1}\right)^2 + \frac{Fl}{D_h} \right]$ $\tau = (2.4 - l) \times 10^{\left(\frac{0.25 + 0.535l^8}{0.05 + l^8} \right)}$

3.4 TRACE range

TRACE could best-estimate the physical phenomena of large-break LOCA and small-break LOCA. However, as a general rule, like the other computational codes, TRACE has its assessment range. The transients that have the following characteristics, are not appropriate for simulation by TRACE, such as too large momentum transfer at a localized level, too large changing asymmetries in core power, and too large viscous stresses which are equal to or larger than wall shear stress [15].

4 Model simulation

The graphical representation of the TRACE-Model using SNAP (Symbolic Nuclear Analysis Package software) is displayed in Figure 4.1. The LSTF has been modeled with 71 hydraulic components (8 BREAKs, 6 FILLS, 40 PIPEs, 2 PUMPs, 1 PRIZER, 11 VALVES and 1 VESSEL). In order to represent the heat transfer processes, 28 Heat Structure components (2 for Steam Generator U-tubes, 12 for core power, 8 for PV and other main pipes heat losses and 3 for lower core plate heat transfer) have been designed. However, some unessential components are emitted to speed up the computation, such as the residual heat removal heat exchanger, jet condensers, break flow storage tank, high pressure injection system.

Figure 4.1 TRACE model of LSTF.

In addition, 63 control systems are built to simulate the complicated control systems and guide the transient scenarios. For example, problem time is set to be a signal variable, when it crosses 100 s, a trip will be set to the state of On(For) by TRACE which activates break valve table. For accident management, the trip is initiated by the signal variables of the temperature at all core exit locations. Most trips are activated when the pressure of vessel lower plenum reaches the

set points of the trips. All of the trips choose the type of On(For) || OFF or OFF || On(For). This kind of trips only have two possible states which are On(For) and OFF. When the status is On(For), the corresponding component-action table will be evaluated. With a status of OFF, the related component-action table is not evaluated. For the value of determining the status, the trips have two set points. They need large enough difference between subranges to avoid oscillating changes.

In order to execute the calculation, the minimum time step is set to be 1.0E-8 for the first 2000 s, and it is reduced to be 1.0E-9 for the rest time until the end of the transient.

4.1 Pressure vessel and its internals

In the present thesis, a 3-D VESSEL component in cylindrical geometry (r, θ, z) is designed to simulate the PWR pressure vessel (PV), as shown in Figure 4.2. The meshes are defined by specifying the radial r , azimuthal θ and axial z coordinates. In the thesis, the VESSEL is nodalized by $5 \times 4 \times 46$ for $r \times \theta \times z$. The VESSEL type uses the default option '[0] RPV' and '[0] No Flow Above or Below'. To model the heating rods and other metal structures, VESSEL component is coupled to HTSTR components and POWER component.

Figure 4.2 Cell nodding for PWR vessel.

As shown in Figure 4.2, there are 46 cells axially for the total height of 10.9574 m. Inside of the PV, it consists of the fuel-assembly reactor core, the lower plenum, the upper plenum, the upper head and the downcomer. All important elevations marked in the experiment are nodalized in different cells to finely represent the experimental phenomena. The first three cells are used to simulate the lower plenum while the 'core plus the end box' is modeled by the

meshes from 4th cell to 33rd cell. The lower core plate is modeled by the 34th cell. The upper plenum is modeled by the meshes from 35th cell to 38th cell. The upper core plate is modeled by the 39th cell which exclude the flow path between the upper head and upper plenum except the ones provided by the eight CRGTs. At last, the upper head which is above the upper core plate is modeled by the meshes from 40th cell to 46th cell. In order to accurately represent the axial core power profile, as shown in Figure 4.3, the core is finely nodalized with ten units. Each unit contains 3 cells whose length is 20 cm, 20 cm and 2 cm respectively. The 2 cm-length cell is used to represent the core spacer whose flow area is much smaller than that of the core.

Figure 4.3 Axial power profile for high-power rods.

Figure 4.4 shows the radial sectioning and distribution of the PV. The inner diameter of PV is 0.64 m and it is divided into 5 rings in radial direction. The innermost three rings represent the lower plenum, the core and the upper plenum from the bottom until 40th axial cell. The core barrel is simulated by the 4th ring and it covers the cells from 1st cell to 40th cell in axial direction. The 5th ring with the axial cells from 1st cell to 40th cell aims to simulate the downcomer. So the 4th ring whose axial flow area is zero could prevent the flow in the core from mixing with the flow of downcomer. The upper head extends to encompass the highest 6 axial with all 5 radial levels.

Figure 4.4 also indicates the radial power distribution. As illustrated in Figure 4.4, the radial power distribution is realized by different numbers of the heating rods distributed to three heating zones in LSTF. The boundaries of the zones are indicated by red, blue and green lines respectively. According to the different peak factors each zone presented, the core is simulated by three rings. The area of each ring equals to the area of corresponding heating zones. For example, the first ring (painted in red) corresponds to the innermost 4 heating zones (21, 22, 23 and 24) whose peak factor is 1.00 and the radius of first ring is calculated by the square root of

the area of the innermost four zones dividing pi. The radius of other core rings are calculated similarly and listed in Table 4.1. In addition, there are four HTSTR components coupled with each ring. In HTSTR components, the power fractions are defined in Powered Components table. For example, the first ring including 180 heating rods could be simulated by 4 HTSTR components with 45 rods each.

Figure 4.4 Radial sectioning and power distribution.

Table 4.1 Radial power distribution.

Ring	Radius [m]	Heat zone	Peak factor	Heated rod number	Heated power fraction
Core: Inner Layer (red)	0.1026	21,22,23,24	1.00	180	0.174947
Core: Middle Layer (blue)	0.1776	13,14,15,16,17,18,19,20	1.435	360	0.5021
Core: Outer Layer (green)	0.257	1,2,3,4,5,6,7,8,9,10,11,12	0.71	468	0.322953

In addition to the radial power distribution, the axial power distribution is also simulated. As shown in Figure 4.3, for different height of heating rod, the power ratio is different. In TRACE model, axial power distribution is realized by initiating the power shape option in which the power shape table could be input by different power ratio values at corresponding locations in POWER component. The power ratio values are listed in table 4.2.

In the model programmed by Gallardo [14] and Freixa [15-16], the vessel is divided by 10 sections in azimuthal direction. In order to speed the computation, four azimuthally subsections are applied in the thesis with 90 degree for each subsection. The PV connections with hot legs, cold legs and control rod guide tubes (CRGT) are modeled approximately in azimuthally sections.

Based on the experimental conditions, the connections between the PV and hot or cold legs should be attached to 37th cell with different radial cells of the 3rd (lower plenum) and 5th ring

(downcomer) respectively. In addition, a leakage between the hot leg (HL) and the downcomer is simulated by a bypass PIPE connected between HL PIPE and the PV 5th ring on the same PV axial level. The leakage is realized by two symmetrical lines and determined by two nozzles whose inner diameter is 3.5 mm. In each line, a hand control valve is located to adjust the leakage mass flow rate which allows 0.2 % bypass flow rate for each loop.

Table 4.2 Axial power distribution.

Elevation (mm)	Length (mm)	Power ratio	
		High-power rod	Low-power rod
0		0.1204	0.1204
409	409	0.3633	0.3628
815	406	0.8135	0.8138
1221	406	1.1736	1.1736
1627	406	1.4071	1.4071
2033	406	1.4945	1.4948
2439	406	1.4071	1.4071
2845	406	1.1756	1.1736
3251	406	0.8135	0.8138
3660	409	0.3633	0.3628
		0.1204	0.1204

Control rod guide tubes (CRGT) which form the flow path between the upper head and the upper plenum are simulated by 8 PIPEs. The flow area is calculated according to the circular area between two rings whose inner diameters are 18 mm and 24 mm respectively. PIPEs are modeled with high K-factor value, since the real geometry of CRGTs is more complex in experiment. In addition, the CRGTs are considered as evenly distributed in four azimuthally subsections.

In VESSEL component, all fluid flow areas and volumes are dimensioned and computed according to the geometric mesh spacing and the volume fraction. The mesh spacing could be specified in Geometry and Connection tables. The fraction can be set up in Volume and Edge data.

In addition, the rod distribution in the core and spacer is also considered to correctly simulate the core flow area. The flow area for free space in the core is calculated by following equation:

$$A_{Core} = \pi \cdot r_{Core}^2 - 1008 \cdot \pi \cdot r_{HR}^2 - 136 \cdot \pi \cdot r_{IR}^2 - 8 \cdot \pi \cdot r_{DP}^2 = 0.1134 \text{ m}^2 \quad (4-1)$$

r_{Core} - the radius of core

r_{HR} - the radius of heated rods

r_{IR} - the radius of instrument rods

r_{DP} -the radius of differential pressure element

The core spacer flow area is calculated by:

$$A_{Corespacer} = A_{Core} - A_{Spacer} = 0.0648 \text{ m}^2 \quad (4-2)$$

Hydraulic diameters are calculated by:

$$H_D = \frac{4A}{P_w} \quad (4-3)$$

H_D -hydraulic diameter

A -flow area

P_w -the wetted perimeter

4.2 Break unit

In the TRACE model, a PIPE component with the same geometry as the experimental condition described in Section 2.3, a VALVE component and a BREAK component are connected with each other to simulate the break unit. In addition, the break trip which is controlled by problem time (when problem time t is equal to 100 s, the trip is activated) is coupled with the VALVE component. Totally, 13 different valve types are provided for simulation in TRACE. For the VALVE used in break simulation, 'Flow Area Table After Trip' type is chosen, because in this type, the valve opening time and size after the break trip could be specified by editing a flow area table. In the LSTF ROSA test 6-1, the break valve is connected to a storage tank where the characteristics of choked point are met, so the choked flow model is activated by choosing default choke in choked flow model. In addition, a detailed break valve open table is defined in the TRACE model to ensure the consistency with the experimental integrated break mass. Because during the experiment, single phase mass flow is determined by measuring differential pressure across a venturi flow meter or a calibrated orifice; the two-phase flow rate is calculated by the values provided by venturi flow meter and gamma-ray densitometers (Figure 2.3) which measure the fluid density. But the two-phase flow measurements are not calibrated which means they are only for qualitative analysis. So to simulate the experimental conditions as close as possible, the integrated mass which is more reliable is considered as the main revision parameter. As it is mentioned in reference [3], the upper head break nozzle elevation is 8500.6 mm which locates at the point between 45th cell and 46th cell in the model, however, the break unit is approximately connected with the upper head at VESSEL 45th cell in the simulation.

4.3 Pressurizer

In the TRACE model, the pressurizer reservoir is simulated by the PRIZER component while the surge lines are modeled by PIPE components. PRIZER could be connected to another 1D hydraulic components at both top and bottom junctions. In the present study, the top junction is connected with a VALVE component and a BREAK component to simulate the relief/safety valves. The bottom of the pressurizer is connected to the surge line, which is modeled by a PIPE component.

However, no relief valves or spray would be actuated in the transient, since the system pressure drops due to the LOCA at the upper head. In this case, the electrical heaters in pressurizer will be activated to produce more steam and increase the pressure. And the heater/sprayer logic is activated to simulate this phenomenon. By activating the logic, the system pressure could maintain the desired pressure specified by the user. For example, it is used to maintain the

pressure of 15.5 MPa in this work. Nonetheless, it cannot simulate the actual sprayer. Actual sprayers should be modeled by PIPE components with side junctions. Its status of off/on and mass flow rate is controlled by a predetermined table. The table would be evaluated once the corresponding trip is triggered. In the model, actual sprayers are omitted here, due to the complex setup and the fact that the pressure does not increase during the simulated transient scenario. But the spray line which could suck some coolant from cold leg to pressurizer is not omitted. It is simulated by a PIPE component. Although the spray water would not be actuated into the pressurizer in the transient, it might inject the coolant to the cold leg for cooling due to the natural circulation. So it is simulated and located besides the top of pressurizer with same geometry, bending and right elevation as the experiment.

4.4 Steam generators (SG)

Steam generators are the components to transfer the heat from the hot reactor coolant to the feedwater in secondary system. Both in primary system and secondary system, there are the components of the SG.

4.4.1 SG primary side

In TRACE, SGs in primary system are simulated by PIPE components by specifying different geometry parameters and orientation inputs for inlet plenum, out plenum and U-tubes. For example, U-tubes are bended to form U-shape. In the TRACE model, the heat exchange between primary system and secondary system is modeled by HTSTR component whose inner edge tied to the U-tubes while outer edge connected to the flow in SG secondary side. There are 141 u-tubes of nine different lengths which made up the tube bundles of LSTF. In the TRACE model simulated by S. Gallardo et al [14], it is claimed that the number of U-tube group does not change the final results significantly. In the present study, only one PIPE component with averaged conditions is used to simulate the U tube bundles by setting the surface multiplier to be 141.

4.4.2 SG secondary side

The SG secondary side consists of the boiler region which exchange heat with U-tubes in the primary side, downcomer pipes, feed water system and the upper region. The upper region includes steam separator, steam dryer, steam line, MSIVs and relief valves and so on. The boiler region is modeled by PIPE component which is coupled by HTSTR components to simulate the heat exchange with the primary system. The downcomer pipes and the feed water injection pipe are simulated by PIPE component. For the upper region, since TRACE 5.0 did not develop any pre-determined steam generator component, the steam separator and dryers are simulated by SPED component as suggested by the TRACE 5.0 user menu [16]. The other secondary system components such as the storage tank and turbine or condenser are simplified by using FILL or BREAK components just with the boundary conditions specified.

4.4.3 SG relief valves

In the experiment, the opening and closing of steam relief valves is controlled by the secondary system pressure values and core exit temperature which triggers the accident management at 623 K. In TRACE simulation, two VALVE components sharing the same geometry and initial conditions are combined to simulate SG RVs. One is activated by accident management trip which uses the core exit temperature as the signal variable while the other is controlled by secondary system pressure. When the signal of secondary system pressure exceeds 7.95MPa, the trip will turn to the status of On(For), and as a result, the valve table of fully opening of RVs will be evaluated. Due to the opening of RVs, the pressure suffers a drop. As soon as it drops down to 7.65 MPa, the trip turn to ON(Rev)which closes the valve. The valves switch off and on alternately until the accident management is activated.

4.5 Other components of primary system

Except PV, SG and pumps, all the other components in the primary system are modeled by PIPEs with same geometry as experimental conditions. For the PIPE type, 'no accumulator' type is chosen. The heat loss is modeled by coupling the PIPEs with unpowered HTSTR components.

The rotational speed of primary coolant pumps is the main controller of primary coolant mass flow. In TRACE simulation, for primary pump type option, the PUMP type of 'table controlled impeller rotation' is used. Corresponding to this type, the speed table is filled with the predefined rotational speed values as the experimental results. When editing the speed table, the trip connected with break valve opening is chosen as the controller trip. As a result, the speed table is evaluated when the break trip is triggered.

4.6 Emergency core cooling system (ECCS)

Emergency core cooling system supplies borated water into the primary system, hot leg, cold leg or directly to the pressure vessel to cool down the core and minimize fuel damage during LOCA. Besides, it also ensure the reactor remain shutdown after the cool down because it could provide the neutron poison. As mentioned in Section 2.3, only accumulators and Low pressure injection systems function normally in ROSA test 6-1.

4.6.1 Accumulator

In TRACE simulation, to accurately reflect the experimental setup, the accumulator system is modeled by three PIPE components and one VALVE component. The accumulator injection line is simulated by 'no accumulator' type PIPE component. The geometry parameters are set according to the data shown in Figure 2.5.

The accumulator tank is modeled by one PIPE component which has to be specified as 'Accumulator type'. Actually, PIPE component type '1' to '3' could be used to simulate the accumulator tank. Take the geometry and later phase gas injection into consideration, '1 accumulator type' is chosen for simulation. Nitrogen gas, locating at the top of the accumulator tank, is defined by activating 'Non-condensable Gas Options' and choosing 'Nitrogen' in the 'Gas and Liquid species' table under 'Model Options'. It is carefully nodalized to make sure the 0.46m^3 non-condensable gas-nitrogen is at the top of the whole tank, below which there is 1.12m^3 water. When defining the initial conditions of accumulator tank, the cells filled with gas have to define the gas volume fraction and 'Non-condensable (NC) Partial Pressure'. In the present model, the stratification between gas and water exists in the 4th cell. So the gas volume fraction of the top 3 cells is 1.0 while the gas volume fraction of 4th cell is 0.499, and the gas volume fraction of rest cells is 0. Besides, the 'NC Partial Pressure' of rest cells is also 0. Such a setting is suggested by reference [16] for accumulator tank.

For a SBLOCA, the depressurization of primary system is relatively slower than a LBLOCA and a periodical pressure rise may occur due to the clearing of loop seal. In the experiment, the accumulators discharge the coolant 3 times for short periods. To allow the accumulator check valve opening and closing alternately, the valve type is set to be 'Flow Area Table After Trip' and the trip setpoint data type should choose 'On(For) || OFF || On(Rev)'. On(Rev) means the closure of the valve, while On(For) means the opening of the check valve. The status of the trip is controlled by the setpoint value '1' to '4'. As Figure 4.5 shown, according to the trip logic, the valve keeps the status of closure when the signal is larger than value S_4 . Until the signal is smaller than S_1 , the valve will be opened and kept this status until the signal is larger than S_4 again which will set the trip to be reverse state to close the valve again. In the model, the pressure difference between accumulator injection line and primary system is set as the signal. The theory for accumulator activation is that when the primary pressure decreased lower than 4.5 MPa. From this, the differential pressure at activation time between accumulator injection line and primary pressure could be deduced and set as the trip S_1 . As soon as the signal variable is small than S_1 , the accumulator trip will be triggered. As a result, the valve of accumulator will be switched on and the coolant will be injected to the primary system driven by gravity and pressure difference.

Figure 4.5 Schematic diagram of trip logic 'On(For) || OFF || On(Rev)'.

4.6.2 Low pressure injection system (LPIS)

In the TRACE model, the low pressure injection system is simulated by a FILL component. The type 'Constant Mass Flow After Trip' is chosen for simulation. The corresponding LPIS trip is simulated by using 'ON (FOR) || OFF' type with the set point 1 equal to 1.23 MPa, since in test 6-1, when the primary pressure drops down to 1.23 MPa, the LPIS will be activated and inject the coolant.

5. Results and discussions

Before initiating the calculations, the model is validated by comparing the geometry input data of TRACE model with the experimental values. Afterwards, steady-state calculations are carried out. After all steady-state parameters are within the error bands of the measurement, the model is run under the transient calculation. The transient calculation results are presented and analyzed in the following sections. At last, the sensitivity study is conducted to investigate the influence of different parameters and setups.

5.1 Model validation: components geometry comparison

As the basic parameters in the fluid-dynamics and heat transfer, the geometry values of the main component are of paramount importance to the experimental results. In order to validate the model built by TRACE code in present study, firstly the simulation geometry values of the main component from the model will be compared with that of experiment. Table 5.1 summarizes the parameters comparison of main component geometry between the simulation and experiment, i.e. volumes. It can be seen from the comparison results that the experimental setup of main components is accurately modeled in the present thesis.

Table 5.1 Geometry parameters comparisons between experiment and TRACE model.

Component type and number in TRACE	Components in LSTF	Volume (m ³)	
		Experiment data	Simulation
Pipe 1	Cold leg	0.1157	0.1157
Pipe 6	Hot leg	0.124	0.1242
Pipe 7	Crossover leg	0.2122	0.2133
Prizer 21	Pressurizer	1.2	1.19766
Pump 8	Main circulation pump	0.0235	0.0235
Vessel 112	Upper head	0.4963	0.4959
	Upper plenum	0.4950	0.4948
	core	0.4477	0.447
	Lower plenum	0.4644	0.4639
	Downcomer+ Core bypass	0.8504	0.85+
Steam generator A	Inlet plenum	0.4351	0.4354
	Outlet plenum	0.2115	0.2114
	U-tubes	0.8384	0.8671
Steam generator B	Inlet plenum	0.4371	0.4354
	Outlet plenum	0.2089	0.2094
	U-tube	0.8384	0.8671

5.2 Steady state results

The simulation results of steady state with TRACE model are shown in Table 5.2, where the experimental initial conditions data are also listed. All steady-state parameters are within the error bands of the measurement. Therefore, it can be believed that the input data is accurate to simulate the test.

Table 5.2 Steady state results comparison between experiment and TRACE model.

Variable	Design Data	Experiment Data	Simulation	Exp. uncertainty
Primary Pressure	15.5 MPa	15.53 MPa	15.62 MPa	± 0.11 MPa
Hot leg fluid temperature	598.1 K	599.1 K	599.8 K	± 2.75 K
Cold leg fluid temp.	562.4 K	563.6 K	563.2 K	± 2.75 K
Mass flow rate (per arm)	24.4 kg/s	24.8 kg/s	24.3 kg/s	± 1.25 kg/s
Main feedwater flow rate	2.76 kg/s	2.74 kg/s	2.8 kg/s	± 0.1 kg/s
Steam Dome Pressure (SG)	7.34 MPa	7.33 MPa	7.26MPa	± 0.054 MPa
Secondary Side Liquid Level	10.7 to 14.28 m	10.23 m	10.94 m	± 0.38 m
Steam flow rate	2.76 kg/s	2.65 kg/s	2.78 kg/s	± 0.1 kg/s
Pumps	188.5 rad/s	85.1 rad/s	85.1 rad/s	± 2.45 rad/s
Downcomer – Upper Head Massflow	0.5 % of core flow	0.3 % (Target Value)	0.27 %	Not measured
Downcomer – Hot LegMassflow	1 % of core flow	0.2 % (= 0.0496 kg/s)	0.6 % (= 0.1485 kg/s)	Not measured

5.3 Transient calculation

In the following sections, transient results calculated by TRACE code are presented and analyzed. It can be seen that the code predicts most phenomena very well through the comparison between TRACE simulation and experimental data. The calculated parameters consist of break unit mass flow, system pressures, steam generator relief valve mass flow, core exit temperature, maximum fuel rod surface temperature, primary loop mass flow rate, collapsed liquid level in the core and upper head and ECCS mass flow.

5.3.1 Chronological sequence

Table 5.3 lists the simulation results of chronological sequence of events during the transient. In order to better compare the simulation with experimental data, all experimental data are kept the same transient initiating time of TRACE modeling, which is corresponding to that in reference [6] of one hundred seconds later.

It can be seen from Table 5.3 that almost all the events in the simulation occur similarly as in the experiment. Discrepancies are kept within a reasonable range except the initiation time of LPIS, which would be discussed later in Section 5.3.9.2. Therefore, it can be concluded that the simulation of control systems is accurate and the TRACE model reproduces the whole experimental process. It is believed that the transient results calculate by TRACE are reliable.

Table 5.3 Chronological sequence of events comparison between experiment and TRACE model.

Event	Experiment (s)	TRACE (s)
Break valve open	100	100
Scram signal	122	121
SI signal	128	127
Break flow from subcooled to two-phase flow	150	147 (gas fraction > 1%)
Primary coolant pump stop	376	380
Break flow to single-phase vapor	800	889 (gas fraction > 90%)
Primary pressure lower than SG pressure	About 900	877
Full opening of SG relief valves	1190	1137
Core protection activated	About 1300	1374
Initiation of accumulator system	About 1400	1440
Initiation of LPIS	About 3000	2003
End of test	3265s	3265s

5.3.2 Break unit mass flow

The simulation results of the break mass flow rate is shown in Figure 5.1, the integrated discharged inventory is illustrated in Figure 5.2, where the black line is the result of experiment and the red one is corresponding to the simulation result of TRACE model.

In Figure 5.1, the mass flow increases sharply after 100 s when the break valve opens and a high mass flow of coolant will be sprayed out due to the large pressure difference between PV and atmosphere. After a short while, the flow rate reaches the critical flow value of more than

12 m/s. After that, it decreases quickly because the break flow switches from single-phase liquid flow to two-phase flow. As there is no HPIS, steam is produced and starts to flow to the upper head due to the fast depressurization and core heating. Hence the upstream break void fraction increases and the break flow changes to be two-phase flow. This two phase flow stabilizes the break mass flow for about 600 s since a balance is built among the core power output, the break loss and the secondary side heat sink. At about 800 s, the mass flow drops down again because the upper plenum is completely emptied. The phase of the flow changed to single-phase vapor. Finally, the mass flow is fluctuated between 0 m/s and 2 m/s. The fluctuations are corresponding to vapor production and condensation due to the coolant injection in PV by ECCS.

As shown in Figure 5.2, the simulation results of integrated coolant mass through break are always close to that of the experimental data. At about 800 s, the total mass is about 3600kg. After that, the rise of the total mass lost is slowed down until the end of test with the value of 5500 kg.

It can be seen from the both figures that the TRACE results are comparable with the experimental values generally.

Figure 5.1 Break mass flow rate.

Figure 5.2 Integrated coolant mass through break.

5.3.3 System pressure

5.3.3.1 Primary system pressure

The primary pressures evolutions are calculated with the TRACE model, the results are plotted in Figure 5.3. The red line describes the simulated evolution while the black one represents the experimental data. When the break valve is switched on, there is a short period of fast depressurization of the primary system. When the break flow changes to two-phase flow, the primary pressure stabilizes at 8 MPa. After that, it decreases again since the break flow alternates to single phase vapor flow. Comparing with the experimental data, the TRACE calculation predicts a quicker drop of pressure at about 900 s, because the corresponding break mass flow is predicted higher (Figure 5.1). During the depressurization, the pressure shows a rebound at 1500 s in the experiment which is around 100 s earlier than the simulation results. Such a difference is due to the different coolant injection process by the ECCS (accumulator injection) between the experiment and simulation, and hence a different time of clearing of loop seal. At time 2000 s, there is a significant deviation between experimental data and TRACE simulation results. The difference decreases to a negligible level at the end of the test, since accumulators are empty of water and LPIS is actuated.

Figure 5.3 Primary pressure.

5.3.3.2 Secondary system pressure

Figure 5.4 shows the simulation result of secondary pressure in red line, the experimental data is also plotted in the figure with black line for comparison. After the break, the secondary pressure starts to rise due to the closure of MSIVs. But as soon as it reaches the value of 7.95 MPa, the pressure of secondary system is maintained within 7.95 MPa and 7.65 MPa by SG relief valves (RVs). At about 1200 s, the secondary side depressurizes gradually because of the initiation of the accident management of the permanent opening of SG RVs. It can be seen from the figure that the TRACE simulation reproduces the whole evolution very well, including the oscillation and slow depressurization.

Figure 5.4 Secondary system pressure.

5.3.4 Steam generator relief valves mass flow

Figure 5.5 plots the mass flow rates of steam generator (SG) relief valves (RVs). Experimental data is presented by the black line and TRACE results are shown in the red solid line. Both have several periods of sharp increase and decrease of the mass flow rates of SG RVs. However, there is minor difference between TRACE prediction and experimental data. It may result from that the experimental data is not calibrated. If considering the initial value of all the experimental data as zero, the integrated value of rest data is almost the same as the TRACE simulated vapor inventory. So the simulation of SG relief valves which is connected to safety management shows reasonable agreement with the experiment.

Figure 5.5 Steam generator relief valves mass flow rates.

5.3.5 Core exit temperature (CET)

CET is an important measurement parameter for the accident management [24]. So in present study, CET is calculated by TRACE code, and the result is shown in Figure 5.6. When the transient is initiated, the CET suffers a rather small drop followed by an increase and finally drops again. The possible reason for the fluctuation is because of the primary coolant mass flow which will be discussed in Section 5.3.7. After that, there is 600 s duration of a relatively steady period for CET. Then it starts rising again at about 1000 s. It is obvious in Figure 5.6 that the TRACE simulation accurately reflects the rising time and trend of CET until 1300 s. But the peak value of the simulation shows slightly higher than that of the experiment. This may due to the thermal-couple measuring point in the experiment locates at the bottom region of corresponding simulation cell. After the peak, the CET suffers a quick drop because the accumulator starts to inject coolant to the system. However, there is a steady phase of about 100 s duration after the first drop since the loop seal is clearing. It corresponds to the rebound of the primary pressure as shown in Figure 5.3. At about 1700 s, the CET drops sharply again. In the TRACE simulation, the second drop of CET is much slower and finally to about 450 K which is 40 K lower than the corresponding experimental data.

In Figure 5.7, the primary pressure and secondary system pressure calculated by the TRACE model are plotted in the same figure, whose comparison could explain the CET rising. It can be seen that the primary pressure is higher than secondary system pressure before 900 s, which ensures of the cooling of the core by natural circulation. However, when the primary pressure

drops below the secondary system pressure due to the fast depressurization of primary system, the vapor in the primary system cannot be condensed any more, and the cooling of the core is ineffective. As a result, the core become uncover and CET begins rising steeply.

As shown in Figures 5.3, 5.4 and 5.6, the pressure prediction of TRACE code show the similar value to the experimental data at the end of test, but the CET prediction at the late phase is different with experimental data. It can be inferred that there might be other reasons for the significant CET difference. In fact, the core is still uncovered in late stage of experiment, while in TRACE simulation it is covered with coolant, which will be discussed in Section 5.3.8.

Figure 5.6 Core exit temperature.

Figure 5.7 Primary and secondary pressures.

5.3.6 Maximum fuel rod surface temperature

The maximum fuel rod surface temperatures are presented in Figure 5.8 with the black line for the experiment and the red line for simulation. The overall trend is similar to CET evolutions. After scram, it drops down a little because of the core power decay. Then the maximum fuel rod stabilizes until at round 1000 s, when it starts to rise quickly to the allowable highest temperature of 958 K. During this period, the discrepancies between the simulation results and the experimental data are kept within a reasonable range of less than 10 K. The calculated maximum fuel rod surface temperature has the similar peak value as that of experiment, but simulated result show a delay comparing with the experimental data, since the predicted core collapsed water level shows a slower decrease than that of experimental data, as shown in Figure 5.12. After that, the temperature decreases because the core protection system is activated and the ECCS also starts to inject coolant. Finally, the temperature decreases down to 410 K at the end of the simulation while the corresponding experiment data is about 450 K.

It can be seen from the figure that TRACE almost reproduces the evolution of the maximum fuel rod temperature especially in the early phase. However, since it spends more time to reach 958 K, so there is a time delay for the activation of the core protection system and the temperature decrease at the late stage. In Figure 5.9, the total reactor power curves are plotted. It can be seen that the power follows the same decay curve in the simulation as that of the experiment until around 1300 s. But the sudden cut down of the power is delayed for about 100 s in the simulation, due to the delay of core protection system between simulated results and experimental data.

Figure 5.8 Maximum fuel rod surface temperature.

Figure 5.9 Reactor total power.

5.3.7 Primary loop mass flow rate

The mass flow evolutions of primary loop A and B calculated by TRACE code are presented in Figures 5.10 and 5.11 respectively. In general, the values and the trend of the mass flow rate through both loops are almost the same. After break valve is switched on, the mass flow increases sharply due to the large pressure difference between the pump and the PV caused by a large pressure drop in the PV. After the scram, the mass flow drops gradually to 0 m/s because of the coast down of pump at about 350 s.

Figure 5.10 Primary mass flow rate in loop A.

Figure 5.11 Primary mass flow rate in loop B.

5.3.8 Collapsed liquid level in the core and upper head

The following Figures 5.12 and 5.13 show the collapsed water level results in the core and upper head. In Figure 5.12, the experiment value for collapsed water level in the core is about 4.6 m while the corresponding TRACE data is about 4 m. After the break, experiment data increased to 6 m while the TRACE result is still about 4 m. The possible reason is that the experimental value of collapsed liquid level is calculated from the pressure difference between the bottom and the top of the core and the pressure difference will be affected by the significant changes of the primary mass flow rate which may be not considered in the experiment calculation. After about 250 s, the water level starts to decrease gradually. The decrease is accelerated at about 900 s when the upper plenum is empty. The experimental collapsed water level starts to rise at about 1500 s, which is reproduced in TRACE although with a time delay of 100 s. Since 2000 s, the collapsed water level in TRACE simulation shows the same value as the steady state due to the ECCS injection. It is found from the experimental data that core is uncovered at the late phase of experiment. The different response of TRACE simulation leads to a lower prediction of the CET and maximum fuel rod surface temperature at the late phase.

Collapsed water level in the upper head is shown in Figure 5.13. The significant difference between TRACE and experimental data at steady state mainly results from the different measurement location. The experiment measures the data between elevation 7834 mm and 9653 mm. Take the fact that the vessel bottom elevations is -2357 mm into consideration, the data is measured at 12.01 m while the whole vessel height in TRACE simulation is 10.9574 m. So it is measured at the very top of vessel which is neglected in TRACE simulation. In general, after the break, the water level suffers a decrease. It is decreased to 0 m at 1500 s. After that, the collapsed water level in upper head increases to the relatively steady value of 1 m and 0.7 m for the simulation and the experiment respectively since 1800 s.

Figure 5.12 Collapsed water level in the core.

Figure 5.13 Collapsed water level in upper head.

5.3.9 ECCS injection mass flow

Due to the assumption of the HPIS failure in the experiment, HPIS is omitted in the simulation. So only the accumulator injection mass flow and LPIS injection mass flow are calculated and presented in the following sections.

5.3.9.1 Accumulator injection mass flow

Figure 5.14 shows the TRACE results of accumulator injection mass flow. During the simulation, the accumulator is activated at around 1400 s with 1 kg/s injection flow rate, which is almost same as that of the experiment. Because of the changes of pressure drop between accumulator system and primary system, the injection flow rate is shaking from 0 kg/s to 3 kg/s before 2200 s. After that, there is little coolant flow out of accumulator since most coolant is already injected to the system. However, the experiment reaches a peak of 6 kg/s at about 1800 s at the second injection. While the TRACE results increases to the highest value of 3 kg/s at about 2000 s. After 2400s, the nitrogen is injected into the system in the experiment [4], but such a phenomenon is not reproduced in TRACE simulation.

For a SBLOCA, the depressurization of primary system is relatively slower than a LBLOCA. There will be periodical pressure rises due to the clearing of loop seal, which will divide the accumulator injection into several short periods. In the experiment, the accumulators discharge the coolant three times for short periods while it injects for more than three times of shorter periods in TRACE modeling. It may be the consequence of different pressure evolutions between TRACE prediction and experimental data. Another possible reason is that the control principle of simulated accumulator valve is different from that of experiment.

Figure 5.14 Accumulator injection mass flow rate.

5.3.9.2 LPIS injection mass flow

The LPIS injection mass flow rates are illustrated in Figure 5.14. The LPIS mass flow calculated by TRACE code increases from 0 kg/s to 1.3 kg/s at about 2000s. While the injection mass flow measured in the experiment is started from 3000 s and finally reaches the value of 1.5 kg/s. It can be seen from Table 5.3 and Figure 5.14, the lower pressure injection system in the TRACE model is activated earlier than that in experiment.

Figure 5.15 LPIS injection flow to CL.

5.4 Sensitivity study

The work is performed to improve the model and investigate the influence of different modeling. The improvement focused on two aspects: the simulation of break unit and leakage between downcomer and hot leg respectively. In addition, the effects of the break location, LPIS simulation and accumulator setup are also studied.

5.4.1 Break unit simulation

It is known that there is no choked flow multiplier for single vapor phase in TRACE code, which may result in some uncertainties during TRACE calculations. Figures 5.16 - 5.19, show the simulation results of break mass flow rate, integrated break mass, CET at center and primary pressure, whose model use default choke for break mass flow and marked as 'TRACE break 1'. It can be seen from the figures that the simulation results show a significant difference after 1200 s comparing with experimental data. To reduce the errors, a modified model is developed by refilling the break valve table with different array of values to control the flow area. The values are set just to follow the experimental evolution of integrated mass flow. The results of the modified model are also added in Figures 5.16 - 5.19 by red line. The principle for this modification is to make the simulated integrated mass flow as close as experimental data, since the measurement of mass flow in the experiment has higher uncertainty value, as discussed in Section 4.2.

As shown in the figures, the model with default choked flow seems over-predicts the break mass flow especially when the coolant is in single phase superheated vapor. The reason may be that appropriate model cannot be actuated in TRACE for the break located at the upper head of PV. In the experiment, the break is above the upper head which could be considered as some kind of off-take geometry for PV. But in TRACE code, the off-take model can be actuated only for the scenario that LOCA is located at a branch pipe while not the one connected with VESSEL component. In addition, the single-phase vapor choking multiplier is not provided to reduce the uncertainties.

Figure 5.16 Break mass flow rate comparisons of different break simulation models.

Figure 5.17 Integrated mass thought break comparisons of different break simulation models.

From Figure 5.18 (the CET prediction) and Figure 5.19 (the primary pressure results), it can be seen that the pressure in the PV simulated by the 'TRACE break 1' model suffers a quicker drop to final steady value with a small rebound at 1400 s and its CET cannot rise to higher value after about 1400 s. The fast depressurization is caused by the high prediction of single-phase vapor loss rate. As a result, the accumulator is activated early and the coolant is injected to the system which prevents the CET from rising.

Figure 5.18 Core exit temperature.

Figure 5.19 PV lower plenum pressure.

5.4.2 Leakage between downcomer and hot leg

In order to study the effect of leakage mass flow area between downcomer and hot leg on the simulation results, different models which manipulate different sets of leakage mass flow area are calculated and shown in Figures 5.20 - 5.22. The only difference among the TRACE models lies on the flow area of the leakage. For the edge flow area equals to $2\text{E-}5 \text{ m}^2$, the model results are presented as the blue line. Other sets include $6\text{E-}5 \text{ m}^2$ and $8\text{E-}5 \text{ m}^2$ whose results are colored by red and green respectively. Black line presents the results of the experiment.

In Figure 5.20, the leakage mass flow rate under various leakage areas has two short increases due to the increasing mass flow of primary loop. After 250 s, the leakage mass flows from the downcomer to the hot leg since the pump speed decreases to zero and the pressure difference drags them out of the PV. With the decrease of water level, the reverse mass flow drops down to 0 m/s at about 900 s. After that, the leakage mass flow fluctuates severely due to the ECCS injection and the temporary clearing of loop seal. Comparing the development of calculated leakage mass flow rate under various leakage flow areas, the one under the leakage flow area of $2\text{E-}5 \text{ m}^2$ is closest to the experimental data at the steady state as shown in Figure 5. 20.

It can be seen from Figures 5.21 & 5.22 that larger flow area could lead to larger drop of pressure and earlier rising of CET. It is reasonable because the larger flow area allows larger coolant mass flow rate leaking from PV and the corresponding pressure will be decreased faster. As a result, it causes the earlier CET rising. Although the leakage between downcomer and hot leg only occupy a rather small fraction of the total core mass flow, it exerts significant influence

on the distribution of primary flow and primary pressure. But the different leakage flow areas do not affect the final steady temperature and pressure. All the predications stabilize at similar values both for CET and pressure.

Figure 5.20 Leakage mass flow between downcomer and hot leg.

Figure 5.21 Core exit temperature.

Figure 5.22 PV lower plenum pressure.

5.4.3 Break location

As mentioned in Section 4.2, the elevation of break nozzle is 8500.6 mm, and it locates at the point between '45th cell 3rd ring' and '46th cell 2nd ring'. So the effect of break location is investigated. In Figures 5.23 - 5.25, the green lines describe the results of the TRACE model whose break connection is at the '46th cell 2nd ring', while the red lines represent the simulation results of the model with the break pipe connection at the '45th cell 3rd ring'. The experimental data marked with black lines are also plotted for comparison.

From the Figure 5.23 - 5.25, it can be seen that the simulated results of the integrated break mass, the CET at center and the primary pressure, which predicted by two different models, have a minor difference. It means that the effect of break location on the integrated break mass, the CET at center and the primary pressure is very small. Therefore, although the simulated break location is not as precise as experimental setup, the result is reliable.

Figure 5.23 Integrated break mass.

Figure 5.24 Core exit temperature.

Figure 5.25 PV lower plenum pressure.

5.4.4 Accumulator simulation

The sensitivity effect of accumulator simulation is studied since it seems to be the main cause of the discrepancies at the late stage. Three different control logics ('time control model', 'no control model' and 'pressure control model') of accumulator injections valve are defined and tested in TRACE code. 'Time control model' means the opening and closing of accumulator valve are triggered at certain time instances. It is predetermined in the flow area table to keep consistent with that of experiment. 'The pressure control model' is that the accumulator is controlled by a trip whose status lay on differential pressure between accumulator system and primary pressure, as described in Section 4.6.1. In the 'no control model', no special controller is added.

The simulated results by different models are shown in Figures 5.26 - 5.28. In the figures, it shows small differences for the three models. In comparison, 'pressure control model' could predict a relatively higher mass flow rate of accumulator injection while the 'no control model' shows the worst prediction. But the data of core exit temperature and primary pressure are similar among three models, as shown in Figures 5.27 - 5.28.

Figure 5.26 Accumulator mass flow.

Figure 5.27 Core exit temperature.

Figure 5.288 PV lower plenum pressure.

Since the pressure control logic shows better simulation result, different groups of the differential pressure set points are modeled and the results are compared in Figure 5.29 - 5.31. As pointed out in Section 2.3, when the primary pressure drops down to 4.5 MPa, the accumulator will be initiated. So the pressure difference set point of valve opening could be deduced. But the control of accumulator valve closure is unknown, so pressure difference between 'set point 1' and 'set point 4' (Figure 4.5) are designed by three values which are named DP large, DP small and DP middle. It can be seen from Figures 5.29 - 5.31 that the results do not show significant difference between different valve closure set points. The peak value may be increased a little by setting small DP, but the calculation cannot execute to the end of the transient under the current time step data. For example, for the DP small, the pressure difference between 'set point 1' and 'set point 4' is 1E4 Pa and the calculation is interrupted at about 2200 s.

Figure 5.29 Accumulator mass flow.

Figure 5.29 Core exit temperature.

Figure 5.30 PV lower plenum pressure.

6. Conclusions

Motivated by validation of TRACE code in simulating the SBLOCA at pressure vessel (PV) upper head, as well as trying to interpret the corresponding transient thermal hydraulic performance, a computational model is developed to simulate the ROSA-V project test 6-1 by using TRACE code 5.0 in this thesis.

Basically, the steady-state and transient-state of test 6-1 are simulated in present study, and the simulation results are compared with the corresponding experimental data to validate the TRACE code. The steady-state presents the initial conditions, while the transient process shows the development of some important parameters during the SBLOCA, such as the break mass flow rate, the pressure in both primary system and secondary system, the core exit temperature (CET), the maximum fuel rod surface temperature, the primary loop mass flow rate, the collapsed liquid level of the core and the emergency core cooling system (ECCS) injection. In addition, the sensitive study is also performed to improve the model and investigate the influence of different modeling for the simulations of the break unit, the leakage between downcomer and hot leg, and the ECCS system. It can be concluded as following:

In general, the simulation results of both the steady-state and the transient-state show good agreement with the experimental data. Discrepancies of chronological sequence are kept within a reasonable range except the initiation time of low pressure injection system (LPIS). It reveals that TRACE model could predict such SBLOCA with reasonable results.

However, at the late phase of simulation, there is a difference of primary pressure prediction due to higher break mass flow predicted in TRACE than that in experiment. It leads to the different coolant injection process by the ECCS (both accumulator and LPIS), which results in a fast depressurization and affects the primary pressure evolution again. In addition, the simulated results of the CET and the maximum fuel rod temperature are also different from experimental data at the late phase, since the core collapsed water level shows a slower decrease than that of experimental data and the core is still uncovered in the experiment at the end of test while it is covered by coolant in TRACE simulation.

From the sensitivity study, it is believed that the accurate prediction of the break mass flow rate and the flow area of the leakage between downcomer and hot leg exert significantly influences on the predicted results the CET and the pressure. When the coolant is in the single phase vapor, TRACE over-predicts the break mass flow for the break located at the upper head of PV and it is difficult to adjust the over-predicted results since the choked flow multiplier is unavailable for the single vapor phase in the code. In addition, the larger flow area between downcomer and hot leg could lead to larger drop of pressure and earlier rising of CET, so if the CET rising is too late, it will be helpful to adjust the leakage flow area. Furthermore, according to simulation results, it is shown that the difference of the break location between the experiment and the simulation at the VESSEL upper head cells due to the limited resolution has negligible effect on the integrated break mass, the CET and the primary pressure. As a result, it

is unnecessary to renodalize the VESSEL component if the break connection does not locate exactly at the same point. Finally, the study of accumulator coolant injection is investigated which demonstrates that neither the different control logics of accumulator injection valves nor the different values of the differential pressure for activating the accumulator injection valves has great effect on the data of core exit temperature and the primary pressure.

References

- [1] P. Bengt, Light water reactor safety. Stockholm: Pergamon Press, 2009.
- [2] B.K. Steven, H. L. Jay, B.K. Thomas, Nuclear Energy Encyclopedia: Science, Technology, and Applications. Wiley press, 2011.
- [3] T. Mura, N. Gun, I. Gen, ROSA-V Group, ROSA-V Large Scale Test Facility (LSTF): System Description for third and fourth simulated fuel assemblies, Japan Atomic Research Institute, 2003.
- [4] The thermo-hydraulic safety research group, Final Data Report of ROSA/LSTF Test 6-1(1.9% Pressure Vessel Upper-head Small Break LOCA Experiment SB-PV-09 in JAEA).Japan Atomic Energy Agency, 2006.
- [5] United States Nuclear Regulatory Commission, Nrc report NUREG/BR-0353, Davis-Besse Reactor Pressure Vessel Head Degradation-Overview, Lessons Learned, and NRC Actions Based on Lessons Learned, 2008.
- [6] The ROSA-V Group, Experimental data of ROSA/LSTF Test 6-1, Japan Atomic Energy Agency, 2006.
- [7] W.M. Ma, E. Bubelis, A. Karbojian, B.R. Sehgal, P. Coddington, Transient experiments from the thermal-hydraulic ADS lead bismuth loop (TALL) and comparative TRAC/AAA analysis, Nuclear Engineering and Design, Vol. 236, pp. 1422–1444, 2006.
- [8] K. Bjorklund, Comparison between RELAP5 and TRACE for modeling different loads on pipe systems during transient conditions, ISSN: 1401-5757, UPTEC F10035, 2010.
- [9] P. Davide, G. Davor, C. Antonio, E. R. Marco, Analysis of different containment models for IRIS small break LOCA, using GOTHIC and RELAP5 code, Nuclear Engineering and Design, Vol. 241, pp. 1152–1164, 2011.
- [10] J. Suchoszek, F. Cadinu, T. Kozlowski, T.N. Dinh, RELAP5 and TRACE codes comparison and validation under steady-state and transient conditions on the basis of NUPEC data, 12th International Topical Meeting on Nuclear Reactor Thermal Hydraulics, NURETH-12, 2007.
- [11] TRACE V5.0 THEORY MANUAL, Field Equations, Solution Methods, and Physical Models, Division of Safety Analysis, U.S. Nuclear Regulatory Commission, 2008.
- [12] W. Wulff, Critical review of conservation equations for two-phase flow in the U.S. NRC TRACE code, Nuclear Engineering and Design, Vol. 241, pp. 4237– 4260, 2011.
- [13] Y.H. Cheng, C.K. Shih, J.R. Wang, H.T. Lin, An investigation of steam–water countercurrent flow model in TRACE, Annals of Nuclear Energy, Vol. 37, pp. 1378–1383, 2010.
- [14] S. Gallardo, V. Abella, G. Verdú, Assessment of TRACE 5.0 against ROSA Test 6-1, Vessel Upper Head SBLOCA, NRC International Agreement Report, NUREG/IA-0245, 2011.
- [15] J. Freixa, A. Manera, Remarks on Consistent Development of Plant Nodalizations: An Example of Application to the ROSA Integral Test Facility. Science and Technology of Nuclear Installations, Hindawi Publishing Corporation, 2011 (doi:10.1155/2012/158617).
- [16] J. Freixa, A. Manera, Analysis of an RPV upper head SBLOCA at the ROSA facility using TRACE. Nuclear Engineering and Design, Vol. 240, pp. 1779–1788, 2010.
- [17] TRACE V5.0 USER'S MANUAL, Volume 2: Modeling Guidelines, U.S. Nuclear Regulatory Commission, 2008.

- [18] T. Siikonen, A study of critical two-phase flow models, Nuclear Engineering and Design, Vol. 73, pp. 293–302, 1982.
- [19] TRACE V5.0 USER'S MANUAL, Volume 1: Input Specification, Division of Safety Analysis, U.S. Nuclear Regulatory Commission, 2008.
- [20] V. H. Ransom and J. A. Trapp, The RELAP5 Choked Flow Model and Application to a Large Scale Flow Test, ANS/ASME/NRC International Topical Meeting on Nuclear Reactor Thermal-Hydraulics (Saratoga Springs, New York), pp. 799–819, 1980.
- [21] I. E. Idelchik, Handbook of hydraulic resistance. Begell House, 3rd edition, 2001.
- [22] B. S. Massey, Mechanics of Fluids. New York: D. Van Nostrand Co, 1968.
- [23] F. M. White, Fluid Mechanics. New York: McGraw-Hill Book Co, 1979.
- [24] S. Oddbjörn, The role of the core exit thermocouples in accident management, presented in OECD-ROSA/LSTF Meeting in Paris, 31 May – 1 June, 2007.