January 2015

Evaluation of the Safety Systems in the Next Generation Boiling Water Reactor

Ling Cheng
Purdue University

Follow this and additional works at: https://docs.lib.purdue.edu/open_access_dissertations

Recommended Citation
https://docs.lib.purdue.edu/open_access_dissertations/1176

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.
This is to certify that the thesis/dissertation prepared

By  Ling Cheng

Entitled
Evaluation of the Safety Systems in the Next Generation Boiling Water Reactor

For the degree of  Doctor of Philosophy

Is approved by the final examining committee:

Dr. Mamoru Ishii  
Chair

Dr. Takashi Hibiki

Dr. Martin Lope-de-Bertodano

Dr. Steven Wereley

To the best of my knowledge and as understood by the student in the Thesis/Dissertation Agreement, Publication Delay, and Certification Disclaimer (Graduate School Form 32), this thesis/dissertation adheres to the provisions of Purdue University’s “Policy of Integrity in Research” and the use of copyright material.

Approved by Major Professor(s): Mamoru Ishii

Approved by: Takashi Hibiki  6/11/2015

Head of the Departmental Graduate Program  Date
EVALUATION OF THE SAFETY SYSTEMS IN THE NEXT GENERATION BOILING WATER REACTOR

A Dissertation
Submitted to the Faculty
of
Purdue University
by
Ling Cheng

In Partial Fulfillment of the Requirements for the Degree of Doctor of Philosophy

August 2015
Purdue University
West Lafayette, Indiana
For my parents.
TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>LIST OF TABLES</th>
<th>v</th>
</tr>
</thead>
<tbody>
<tr>
<td>LIST OF FIGURES</td>
<td>vi</td>
</tr>
<tr>
<td>LIST OF SYMBOLS</td>
<td>x</td>
</tr>
<tr>
<td>LIST OF ABBREVIATIONS</td>
<td>xii</td>
</tr>
<tr>
<td>ABSTRACT</td>
<td>xiv</td>
</tr>
<tr>
<td>CHAPTER 1. INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>1.1 Motivations of Study</td>
<td>1</td>
</tr>
<tr>
<td>1.2 Background</td>
<td>2</td>
</tr>
<tr>
<td>1.2.1 Safety Systems of the Simplified Boiling Water Reactor (SBWR)[1]</td>
<td>2</td>
</tr>
<tr>
<td>1.2.2 PUMA Facility[3]</td>
<td>7</td>
</tr>
<tr>
<td>1.3 Suppression Pool Mixing and Condensation</td>
<td>12</td>
</tr>
<tr>
<td>1.4 Objectives and Scope of Study</td>
<td>19</td>
</tr>
<tr>
<td>CHAPTER 2. MAIN STEAM LINE BREAK TEST IN THE PUMA FACILITY</td>
<td>21</td>
</tr>
<tr>
<td>2.1 Main Steam Line Break Accident in SBWR</td>
<td>21</td>
</tr>
<tr>
<td>2.2 Initial Conditions for the PUMA MSLB Test[27]</td>
<td>24</td>
</tr>
<tr>
<td>2.3 Initialization Procedure for the PUMA MSLB Test</td>
<td>27</td>
</tr>
<tr>
<td>2.4 Decay Power for the PUMA MSLB Test</td>
<td>29</td>
</tr>
<tr>
<td>2.5 General Discussion of the PUMA MSLB Test</td>
<td>32</td>
</tr>
<tr>
<td>CHAPTER 3 RELAP5 SIMULATION FOR THE MSLB LOCA</td>
<td>37</td>
</tr>
<tr>
<td>3.1 Evaluating MSLB by Utilizing RELAP5 Code</td>
<td>37</td>
</tr>
<tr>
<td>3.2 RELAP5 Simulation for the PUMA MSLB Test</td>
<td>38</td>
</tr>
<tr>
<td>3.3 Scaled-up PUMA Test Data Compared with SBWR Simulation</td>
<td>45</td>
</tr>
</tbody>
</table>
### CHAPTER 4. SUPPRESSION POOL CONденSATION AND MIXING TESTS. 49

4.1 Separated-effect Test on SP Condensation and Mixing .............................................. 49
   4.1.1 SP Separated-effect Test Facility[29] ........................................................................ 49
   4.1.2 Instrumentation[29] ................................................................................................ 53
   4.1.3 SP Separated-effect Test Matrix[29] ....................................................................... 57
   4.1.4 SP Separated-effect Test Results[29] ..................................................................... 61

4.2 RELAP5 Simulation for the Suppression Pool Mixing and Condensation ............ 72

### CHAPTER 5. SUPPRESSION POOL MIXING ANALYSIS ............................................. 74

5.1 Modeling Method ........................................................................................................... 74
5.2 Governing Equations for Turbulent Buoyant Jets and Plumes .............................. 74
5.3 Integral Modeling of the Turbulent Buoyant Jets ....................................................... 80
5.4 Modeling Mixing inside Suppression Pool with Pure Steam Injection ................ 84
5.5 Modeling Mixing inside Suppression Pool with Air Injection ................................. 93

### CHAPTER 6. NUMERICAL MODELING OF THE PUMA SP TEST ............................. 95

6.1 Transport Equations and Turbulence Models in FLUENT[41] .................................. 96
6.2 PUMA Suppression Pool Mixing Calculation in FLUENT ....................................... 99

### CHAPTER 7. CONCLUSION ......................................................................................... 106
7.1 Summary of the Current Study .................................................................................... 106

### LIST OF REFERENCES ............................................................................................... 109

### APPENDICES

Appendix A RELAP5 Nodalization for SBWR MSLB ....................................................... 116
Appendix B Modeling for Suppression Pool Mixing in Matlab .................................... 123
Appendix C User-defined Function in CFD Calculation ................................................. 130

VITA ................................................................................................................................. 132
# LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table 1.1</td>
<td>ADS Valve Actuation Sequence in SBWR</td>
<td>3</td>
</tr>
<tr>
<td>Table 1.2</td>
<td>Scaling Ratio of the PUMA Facility&lt;sup&gt;3&lt;/sup&gt;</td>
<td>8</td>
</tr>
<tr>
<td>Table 2.1</td>
<td>Initial Conditions for PUMA MSLB Test</td>
<td>26</td>
</tr>
<tr>
<td>Table 2.2</td>
<td>PUMA Facility Status after the Initialization Process for MSLB Test</td>
<td>29</td>
</tr>
<tr>
<td>Table 2.3</td>
<td>Decay Power Curve for the PUMA Facility</td>
<td>30</td>
</tr>
<tr>
<td>Table 4.1</td>
<td>Instrumentation Used in the SP Separated-effect Test</td>
<td>54</td>
</tr>
<tr>
<td>Table 4.2</td>
<td>Air Mass Flow Controller Specifications</td>
<td>55</td>
</tr>
<tr>
<td>Table 4.3</td>
<td>Test Matrix for the Vent Opening at the 2&lt;sup&gt;nd&lt;/sup&gt; Level</td>
<td>58</td>
</tr>
<tr>
<td>Table 4.4</td>
<td>Test Matrix for the Vent Opening at the 2&lt;sup&gt;nd&lt;/sup&gt; Level</td>
<td>59</td>
</tr>
<tr>
<td>Table 6.1</td>
<td>CFD Calculation Conditions</td>
<td>101</td>
</tr>
</tbody>
</table>
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 1.1</td>
<td>Schematics of SBWR\textsuperscript{[1]}</td>
<td>5</td>
</tr>
<tr>
<td>Figure 1.2</td>
<td>SBWR Passive Safety Systems\textsuperscript{[1]}</td>
<td>6</td>
</tr>
<tr>
<td>Figure 1.3</td>
<td>Schematics of SBWR ADS\textsuperscript{[1]}</td>
<td>6</td>
</tr>
<tr>
<td>Figure 1.4</td>
<td>Three Level Scaling Methodology Flow Chart\textsuperscript{[3]}</td>
<td>11</td>
</tr>
<tr>
<td>Figure 1.5</td>
<td>3D View of PUMA Facility\textsuperscript{[3]}</td>
<td>12</td>
</tr>
<tr>
<td>Figure 2.1</td>
<td>Three Phases during the Main Steam Line Break Accident\textsuperscript{[2]}</td>
<td>24</td>
</tr>
<tr>
<td>Figure 2.2</td>
<td>Fuel Rod Temperature Profile at 1030 kPa and at Long Term</td>
<td>31</td>
</tr>
<tr>
<td>Figure 2.3</td>
<td>Pressure Trend during the PUMA MSLB Test</td>
<td>32</td>
</tr>
<tr>
<td>Figure 2.4</td>
<td>Enlarged Pressure Trend during the PUMA MSLB Test</td>
<td>33</td>
</tr>
<tr>
<td>Figure 2.5</td>
<td>RPV, DW and GD Water Level during the PUMA MSLB Test</td>
<td>34</td>
</tr>
<tr>
<td>Figure 2.6</td>
<td>GDCS A Injection Flow Rate during the PUMA MSLB Test</td>
<td>34</td>
</tr>
<tr>
<td>Figure 2.7</td>
<td>ADS Actuation Sequences during the PUMA MSLB Test</td>
<td>36</td>
</tr>
<tr>
<td>Figure 2.8</td>
<td>ICS and PCCS Performance during the PUMA MSLB Test</td>
<td>36</td>
</tr>
<tr>
<td>Figure 3.1</td>
<td>Comparison for RPV Pressure</td>
<td>40</td>
</tr>
<tr>
<td>Figure 3.2</td>
<td>Comparison for Break Flow Rate</td>
<td>41</td>
</tr>
<tr>
<td>Figure 3.3</td>
<td>Comparison for RPV Collapsed Water Level</td>
<td>42</td>
</tr>
<tr>
<td>Figure 3.4</td>
<td>Comparison for Lower DW Water Level</td>
<td>43</td>
</tr>
<tr>
<td>Figure 3.5</td>
<td>Comparison for GDCS A Injection Flow Rate</td>
<td>43</td>
</tr>
</tbody>
</table>
Figure 3.6 Comparison for DW Pressure ................................................................. 44
Figure 3.7 RELAP5 Predicted ICS and PCCS Performance ................................ 44
Figure 3.8 Comparison of RPV Pressure in SBWR MSLB ................................. 46
Figure 3.9 Comparison of DW Pressure in SBWR MSLB ................................. 47
Figure 3.10 Comparison of RPV Collapsed Water Level in SBWR MSLB .......... 47
Figure 3.11 Comparison of GDCS A Injection Flow Rate in SBWR MSLB ......... 48
Figure 4.1 SP Separated-effect Test Facility ......................................................... 52
Figure 4.2 Vent Opening at the Second Level .................................................... 52
Figure 4.3 Vent Side Thermocouple Locations ................................................... 56
Figure 4.4 Calculated Heater Power from Measured Steam Flow Rate .............. 56
Figure 4.5 Calibrating Air Line Vortex Flow Meter with Mass Flow Controller .... 57
Figure 4.6 RELAP5 SBWR600 Simulation for SP Separated-effect Test .......... 60
Figure 4.7 RPV, DW and SP Pressure during the LSF1 Test ............................. 64
Figure 4.8 RPV Heater Power during the LSF1 Test ........................................... 64
Figure 4.9 Effect of Initial Pool Temperature on SP Thermal Stratification ...... 65
Figure 4.10 Effect of DW Pressure on SP Thermal Stratification ....................... 65
Figure 4.11 Effect of Steam Flow Rate on SP Thermal Stratification ................. 66
Figure 4.12 Effect of Vent Opening Submergence Depth on SP Thermal Stratification ... 66
Figure 4.13 Pool Temperature at R = 106 cm for 0.07 kg/s Pure Steam Injection Case .. 67
Figure 4.14 2-D Temperature Distributions during Pure Steam Injection .......... 68
Figure 4.15 Schematic of Pool Circulation Driven by Thermal Plume .............. 69
Figure 4.16 Pool Temperature at R = 106 cm for 0.07 kg/s Steam and 3.684 g/s Air ..... 69
<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 4.17 2-D Temperature Distributions during Steam and High Noncondensable Gas Injection Case</td>
<td>70</td>
</tr>
<tr>
<td>Figure 4.18 Schematic of Pool Circulation Driven by Thermal and Air Bubble Plume</td>
<td>71</td>
</tr>
<tr>
<td>Figure 4.19 Nodalization of RELAP5 for PUMA Mixing and Condensation</td>
<td>72</td>
</tr>
<tr>
<td>Figure 4.20 RELAP5 Predicted Temperature Profile for Pure Steam Injection Case (Steam Flow Rate = 0.1 kg/s)</td>
<td>73</td>
</tr>
<tr>
<td>Figure 4.21 RELAP5 Predicted Temperature Profile for Steam/Air Injection Case (Steam Flow Rate = 0.1 kg/s, Air Concentration 5%)</td>
<td>73</td>
</tr>
<tr>
<td>Figure 5.1 Turbulent Buoyant Jet/Plume Flow&lt;sup&gt;[30]&lt;/sup&gt;</td>
<td>77</td>
</tr>
<tr>
<td>Figure 5.2 Test Image of Injection Port (Pure Steam Injection Case)</td>
<td>85</td>
</tr>
<tr>
<td>Figure 5.3 Calculation Domain for the Pure Steam Injection Case</td>
<td>85</td>
</tr>
<tr>
<td>Figure 5.4 One-dimensional Integral Model for Buoyant Jet/Plume</td>
<td>86</td>
</tr>
<tr>
<td>Figure 5.5 Prediction Results vs. Test Data for Steam Injection as 0.07 kg/s</td>
<td>91</td>
</tr>
<tr>
<td>Figure 5.6 Prediction Results vs. Test Data for Steam Injection Rate as 0.12 kg/s</td>
<td>92</td>
</tr>
<tr>
<td>Figure 5.7 Test Image of Injection Port (Steam/Air Injection Case)</td>
<td>93</td>
</tr>
<tr>
<td>Figure 6.1 SP Modeling in FLUENT</td>
<td>102</td>
</tr>
<tr>
<td>Figure 6.2 Initial Status for LSF1 and LSF4 Calculation</td>
<td>102</td>
</tr>
<tr>
<td>Figure 6.3 Pool Temperature in LSF1 Calculation</td>
<td>103</td>
</tr>
<tr>
<td>Figure 6.4 Pool Velocity in LSF1 Calculation</td>
<td>104</td>
</tr>
<tr>
<td>Figure 6.5 Pool Velocity in LSF4 Calculation</td>
<td>104</td>
</tr>
<tr>
<td>Appendix Figure</td>
<td></td>
</tr>
<tr>
<td>Figure A.1 RELAP5 Nodalization for SBWR RPV</td>
<td>116</td>
</tr>
<tr>
<td>Figure</td>
<td>RELAP5 Nodalization for SBWR Containment and Vertical Vent</td>
</tr>
<tr>
<td>--------</td>
<td>----------------------------------------------------------</td>
</tr>
<tr>
<td>A.2</td>
<td>Page 117</td>
</tr>
<tr>
<td>A.3</td>
<td>RELAP5 Nodalization for SBWR PCCS</td>
</tr>
<tr>
<td>A.4</td>
<td>RELAP5 Nodalization for SBWR ICS</td>
</tr>
<tr>
<td>A.5</td>
<td>RELAP5 Nodalization for SBWR GDCS</td>
</tr>
<tr>
<td>A.6</td>
<td>RELAP5 Nodalization for SBWR ADS and MSL</td>
</tr>
<tr>
<td>A.7</td>
<td>RELAP5 Nodalization for SBWR GDCS Equalization Lines</td>
</tr>
<tr>
<td></td>
<td>Page 118</td>
</tr>
<tr>
<td></td>
<td>Page 119</td>
</tr>
<tr>
<td></td>
<td>Page 120</td>
</tr>
<tr>
<td></td>
<td>Page 121</td>
</tr>
<tr>
<td></td>
<td>Page 122</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>---------------------------</td>
</tr>
<tr>
<td>$\delta$</td>
<td>Half width of the jet flow</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Dynamic viscosity</td>
</tr>
<tr>
<td>$A$</td>
<td>Area</td>
</tr>
<tr>
<td>$C_p$</td>
<td>Heat Capacity</td>
</tr>
<tr>
<td>$D$</td>
<td>Diameter</td>
</tr>
<tr>
<td>$Fr$</td>
<td>Froude number</td>
</tr>
<tr>
<td>$g$</td>
<td>Gravity Constant</td>
</tr>
<tr>
<td>$G$</td>
<td>Mass flux</td>
</tr>
<tr>
<td>$Gr$</td>
<td>Grashof number</td>
</tr>
<tr>
<td>$h$</td>
<td>Heat transfer coefficient</td>
</tr>
<tr>
<td>$K$</td>
<td>Form loss coefficient</td>
</tr>
<tr>
<td>$L$</td>
<td>Length</td>
</tr>
<tr>
<td>$m$</td>
<td>Mass</td>
</tr>
<tr>
<td>$Nu$</td>
<td>Nusselt Number</td>
</tr>
<tr>
<td>$q$</td>
<td>Heat flux</td>
</tr>
<tr>
<td>$R$</td>
<td>Radius</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>----------------------</td>
</tr>
<tr>
<td>$Ri$</td>
<td>Rechardson number</td>
</tr>
<tr>
<td>$T$</td>
<td>Temperature</td>
</tr>
<tr>
<td>$U, u$</td>
<td>Velocity</td>
</tr>
</tbody>
</table>

**LIST OF SUBSCRIPTS**

- $0$: Reference point
- $a$: Original point
- $imp$: Adjacent
- $jet$: Impingement
- $m$: Jet flow region
- $m$: Mass
- $r$: Radius
# LIST OF ABBREVIATIONS

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ADS</td>
<td>Automatic Depressurization System</td>
</tr>
<tr>
<td>BWR</td>
<td>Boiling Water Reactor</td>
</tr>
<tr>
<td>DPV</td>
<td>Depressurization Valve</td>
</tr>
<tr>
<td>DW</td>
<td>Drywell</td>
</tr>
<tr>
<td>GDCS</td>
<td>Gravity Driven Cooling System</td>
</tr>
<tr>
<td>GE</td>
<td>General Electric</td>
</tr>
<tr>
<td>HV</td>
<td>Horizontal Vent</td>
</tr>
<tr>
<td>ICS</td>
<td>Isolation Condensation system</td>
</tr>
<tr>
<td>LOCA</td>
<td>Loss of Coolant Accident</td>
</tr>
<tr>
<td>MSL</td>
<td>Main Steam Line</td>
</tr>
<tr>
<td>MSLB</td>
<td>Main Steam Line Break</td>
</tr>
<tr>
<td>NWL</td>
<td>Normal Water Level</td>
</tr>
<tr>
<td>PCCS</td>
<td>Passive Containment Cooling System</td>
</tr>
<tr>
<td>PUMA</td>
<td>Purdue University Multi-Dimensional Test Assembly</td>
</tr>
<tr>
<td>RPV</td>
<td>Reactor Pressure Vessel</td>
</tr>
<tr>
<td>SBWR</td>
<td>Simplified Boiling Water Reactor</td>
</tr>
<tr>
<td>Abbreviation</td>
<td>Description</td>
</tr>
<tr>
<td>--------------</td>
<td>----------------------</td>
</tr>
<tr>
<td><em>SP</em></td>
<td>Suppression Pool</td>
</tr>
<tr>
<td><em>SRV</em></td>
<td>Safety Relief Valve</td>
</tr>
<tr>
<td><em>SSAR</em></td>
<td>Standard Safety Analysis Report</td>
</tr>
<tr>
<td><em>TAF</em></td>
<td>Top of Active Fuel</td>
</tr>
<tr>
<td><em>VV</em></td>
<td>Vertical Vent</td>
</tr>
</tbody>
</table>
ABSTRACT


The thesis evaluates the safety systems in the next generation boiling water reactor by analyzing the main steam line break loss of coolant accident performed in the Purdue university multi-dimensional test assembly. RELAP5 code simulations, both for the PUMA MSLB case and for the SBWR MSLB case have been utilized to compare with the experiment data. The comparison shows that RELAP5 is capable to perform the safety analysis for SBWR. The comparison also validates the three-level scaling methodology applied to the design of the PUMA facility.

The PUMA suppression pool mixing and condensation test data have been studied to give the detailed understanding on this important local phenomenon. A simple one dimensional integral model, which can reasonably simulate the mixing process inside suppression pool have been developed and the comparison between the model prediction and the experiment data demonstrates the model can be utilized for analyzing the suppression pool mixing process.
CHAPTER 1. INTRODUCTION

1.1 Motivations of Study

Design features proposed for the next generation boiling water reactor\(^1\) include the use of passive safety systems, such as gravity driven emergency core cooling system and natural circulation decay heat removal system, to improve the reliability. The performance of these safety systems under a loss of coolant accident (LOCA) should be evaluated before the real commercial deployment of such new reactor design. This can be done by carrying out experiment study on a well scaled down test facility that can reproduce major phenomena encountered during the LOCA of the next generation boiling water reactor. The analysis of the experiment data generated in such facility can help us to further understand the interactions between the safety systems under LOCA conditions. The analysis of the experiment data can also help us to validate the reactor system safety analysis code, such as RELAP5.

Important local phenomenon, such as suppression pool condensation and mixing which affects the overall behaviors of the reactor safety systems, should be addressed more carefully. Separated-effect tests with well controlled initial and boundary conditions should be performed to assess such phenomenon. Analytical model developed based on the test data can be cooperated into the system safety analysis code so that the code capabilities will be improved.
1.2 Background

1.2.1 Safety Systems of the Simplified Boiling Water Reactor (SBWR)[1]

The Simplified Boiling Water Reactor designed by the General Electrical utilizes proven techniques and passive systems to improve the reactor safety and reduce the possibility of core melt down caused by the human error. The SBWR uses natural circulation to transfer the energy released from the core during the normal operation or under accident conditions. The elimination of the outside recirculation pumps and the corresponding connection pipe lines reduces the possibility of the pipe break below the reactor core section. The SBWR emergency core cooling systems (ECCS) and containment cooling systems solely depend on the natural forces or phenomena, such as natural circulation, gravity driven flow or condensation/convection heat transfer, to accomplish their designed safety functions. Comparing with traditional Boiling Water Reactors (BWRs), the SBWR designs are special in the following aspects: 1). No recirculation pumps to drive the coolant flow in the vessel; 2). Low volumetric heat generation rates in the reactor core section; 3). No emergency AC power and no pumps requisition in the design base accidents (DBAs); 4). No operator intervention to active the safety systems in the DBAs. Figure 1.1 shows the schematics of the SBWR.

Safety systems in SBWR include[2]:

The automatic depressurization system (ADS). The ADS consists of eight safety relief valves (SRVs), six depressurization valves (DPVs) and the associated pipe lines, instrumentation and controls. Figure 1.2 shows the ADS schematic in the SBWR. The SRVs will discharge steam from reactor vessel to the suppression pool, while the DPVs will discharge steam from reactor vessel to the upper drywell. The function of the ADS is
to systematically depressurize the reactor vessel in the event of loss of coolant accident (LOCA) or other transients in order to 1). utilize controlled flashing to cooling down the reactor core and send the decay heat from reactor core to containment; 2). allow gravity driven cooling system (GDCS) injects coolant into the reactor vessel; and 3). minimize the mechanic loads to the bases of the reactor vessel generated by the steam blowdown. The ADS is activated when the low water level (Level 1) signal persists for at least 10 seconds. After that, the valve actuation sequences are summarized in Table 1.1.

<table>
<thead>
<tr>
<th>Valve in the GDCS Injection Line</th>
<th>L1 + 0.0 second</th>
</tr>
</thead>
<tbody>
<tr>
<td>Valve in the GDCS Equalization Line</td>
<td>L1 + 150.0 second</td>
</tr>
</tbody>
</table>

The gravity driven cooling system (GDCS). The GDCS provides short-term inventory injection from three water tanks to the reactor vessel after the ADS depressurizes the vessel. The GDCS also provides long-term inventory injection from suppression pool to the reactor vessel to meet the post-LOCA core cooling requirements. The GDCS water tanks are located at upper drywell, above the reactor core regime. The GDCS short-term injection will be initiated 150 seconds after the activation of the ADS. The GDCS long-term
injection will be initiated when the RPV inventory decreases to 1 m above the top of active fuel (TAF) and 30 minutes have been passed since the ADS has been activated. The check valve installed in each GDCS injection pipe line ensures that coolant can only flow into the reactor vessel.

The isolation condenser system (ICS). The ICS directly removes the energy from reactor vessel to outside of the containment through natural circulation. The ICS consists of three independent high-pressure loops, each of which contains a steam supply line, an isolation condenser, a condensate drain line and a noncondensable gas venting line. The isolation condenser is a vertical heat exchanger submerged inside the ICS/PCCS pool. Once ICS is activated, steam coming from the reactor vessel enters the isolation condenser, condenses inside the vertical tubes. The condensate returns to the reactor vessel through the condensate drain line. Eventually, the ICS/PCCS pool will be boiled up by the energy discharged through the isolation condenser. The evaporated steam is vented to the atmosphere. The ICS is activated when the reactor vessel water level falls below Level 2 during the LOCA transient. The noncondensable gas purging through the ICS venting line is operated manually by the operator.

The passive containment cooling system (PCCS). The PCCS provides containment cooling for a minimum of 72 hours after a LOCA. Similar to the ICS, the PCCS consists of three independent loops, each of which contains a steam supply line, a PCCS condenser, a condensate drain line and a noncondensable gas vent line. The PCCS condenser is also a vertical heat exchanger submerged inside the ICS/PCCS pool. The PCCS is a complete passive system. There are no power actuated valves or other component that must be activated for the PCCS to work. During a LOCA, the PCCS takes steam from the upper
drywell, condenses the steam inside the condenser tubes, and then drains the condensate to the GDCS tank. The noncondensable gas accumulated inside the condenser tube will be automatically purged into suppression pool by the pressure difference between the drywell and suppression pool.

The suppression pool (SP). The suppression pool is a large annular chamber that surrounds the reactor vessel which can 1). quench the steam injected through the SRV and the horizontal vent during the blowdown phase; 2). provide long term coolant injection to the reactor vessel through the GD equalization lines. The gas space of the suppression chamber serves as the LOCA blowdown gas reservoir for the drywell noncondensable gas.

Figure 1.1 Schematics of SBWR[1]
Figure 1.2 SBWR Passive Safety Systems\textsuperscript{[1]}

Figure 1.3 Schematics of SBWR ADS\textsuperscript{[1]}
1.2.2 PUMA Facility\textsuperscript{[3]}

The Purdue university multi-dimensional integral test assembly (PUMA) is a well-scaling down test facility to simulate the LOCAs or other transients of SBWR after the RPV depressurized below 1.03 MPa. PUMA facility design follows the three level scaling approach\textsuperscript{[4]} developed by Ishii et al. The first level of scaling is based on the well-established approach obtained from the integral response function, namely, the integral scaling. This first level scaling ensures that the steady states as well as dynamic characteristics of the test facility are scaled down properly. The second level of scaling focuses on the boundary flow of mass and energy between components. This level scaling ensures that the flow and inventory are scaled correctly. The third level scales down key local phenomena and constitutive relations from prototype plant to test facility. The flow chart for three level scaling methodology is shown in Figure 1.3. The scaling ratios from PUMA to SBWR have been summarized in Table 1.2. Compare to the SBWR, PUMA has the height ratio of 1/4, the diameter ratio of 1/10, and the power ratio of 1/200. The time ratio from PUMA to SBWR is 1/2, which means everything happens in the PUMA will be twice faster than in the SBWR. Figure 1.4 shows the 3D view of the PUMA facility.

As can be seen from the Figure 1.4, the PUMA facility mainly consists of four large pressure tanks and two pools open to the atmosphere to simulate various SBWR components. One of them represents the RPV, which will supply steam source during the integral test. The lower part of RPV simulates the core region, where 38 electronic heater rods have been inserted from the bottom of RPV. Three silicon controlled rectifier power controller have been installed to set the heat rod power. Thus, history of the core decay power can be simulated through the computer controller electronic heater rods with
maximum power capability of 400 kW. The chimney, steam separator and dryer components have been installed above the core regime to enable the natural circulation inside the RPV. The RPV is connected to the drywell through two main steam lines (MSLs) and two depressurization valve lines (DPV lines) at the top. If it is necessary, the RPV can also connect to the drywell through the control rod driven (CRD) break line and the reactor water clean-up/shut-down cooling (RWCU) break line at the bottom. The RPV is connected to other components through three ICS steam supply lines and three ICS condensate drain lines, three GD drain lines, and three GD equalization lines.

Table 1.2 Scaling Ratio of the PUMA Facility\cite{3}

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Scaling Ratio (PUMA/SBWR)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Height</td>
<td>1:4</td>
</tr>
<tr>
<td>Diameter</td>
<td>1:10</td>
</tr>
<tr>
<td>Area</td>
<td>1:100</td>
</tr>
<tr>
<td>Volume</td>
<td>1:400</td>
</tr>
<tr>
<td>Pressure</td>
<td>1:1</td>
</tr>
<tr>
<td>Power</td>
<td>1:200</td>
</tr>
<tr>
<td>Time</td>
<td>1:2</td>
</tr>
</tbody>
</table>

The vessel in the dumbbell shape represents the drywell. The PUMA drywell is divided into an upper drywell space and a lower drywell space by an orifice plate, which simulates the reactor vessel support skirt in the SBWR. Except to the RPV, the drywell is also connected to other components through three PCCS steam supply lines, three vacuum break lines, the vertical vent line and various break lines. In the integral test, the lower drywell
collects water/condensate that enters the containment through break line or ADS lines, while the upper drywell usually is filled with steam released from reactor vessel.

The small vessel locates at the high elevation represents the GDCS tank. The gas space of the GDCS tank is opened to the drywell. The GDCS tank is connected to the reactor vessel through three GD drain lines. A check valve has been installed in each GD drain line to ensure that coolant can only be injected from GDCS tank to the reactor vessel. GDCS tank also receive condensate from PCCS through three PCCS condensate drain lines.

The largest vessel represents the suppression pool. A 14 inch diameter vertical vent pipe line connects the PUMA drywell and SP, on which nine 175 mm × 22 mm horizontal vent openings are located at three different elevations under the water surface. The SP is also connected to the drywell through three vacuum breaker lines at the gas space. The vacuum breaker lines will open whenever the SP pressure is larger than the drywell pressure. The suppression pool is connected to the ICS and PCCS through three ICS vent lines and three PCCS vent lines, which are submerged 0.2 m under the SP nominal water level. The noncondensable gas accumulated inside the PCCS condenser tubes will automatically be purged into the suppression pool whenever the pressure difference between the drywell and suppression pool can overcome the vent line submergence hydraulic head. This purging function is important for the operation of the PCCS since the noncondensable gas will seriously decrease the condensation heat transfer rate inside the PCCS condenser tubes. The suppression pool also connects to the reactor vessel through three GD equalization lines.

ICS and PCCS system of the PUMA facility are exactly scaled down from the prototype SBWR except for that the ICS/PCCS pool is much smaller than the pool of the ideal scaling
facility. This distortion can be compensated by filling the ICS/PCCS pool during the experiment once the pool water level drops to a certain value. This action will not affect too much on the ICS/PCCS behavior due to the fact that the ICS/PCCS total heat transfer coefficient is not a strong function of the pool side inventory as long as the condenser is immersed inside the water.
Figure 1.4 Three Level Scaling Methodology Flow Chart[3]
1.3 Suppression Pool Mixing and Condensation

Suppression pool (SP) is part of the reactor containment system. The major function of a reactor containment system is to protect the environment from an uncontrolled release
of radioactive materials in an accident such as a LOCA. This objective is achieved by designing containment system to accommodate all combinations of loads generated by the mass and energy release associated with reactor blowdown from a LOCA. Suppression pool serves as a heat sink to absorb reactor blowdown energy\textsuperscript{[1]}.

The long-term post-accident pressure in the containment is determined by noncondensable gas pressure and steam pressure in the suppression pool gas space\textsuperscript{[5]}. The SP surface temperature, which affects the vapor partial pressure, is important to the overall containment pressure. Next generation reactor tends to use more passive safety systems to reduce cost and improve maintainability and reliability, and there is no mechanism to promote pool mixing. Therefore, realistic modeling of the thermal stratification in the SP during a LOCA is essential for the reactor system simulation codes such as RELAP5 or TRACE to evaluate the containment pressure.

Many works have been published in the past on various aspects of the SP behavior. General Electric conducted a series of tests that examined the SP behavior during the blowdown period of a LOCA. The tests were performed at the pressure suppression test facility (PSTF). These tests provided data on SP behavior at a variety of scales – full scale\textsuperscript{[6]}, one-third area scale\textsuperscript{[7]} and one-ninth area scale\textsuperscript{[8]}. However, the primary emphasis of these tests was on WW mechanical loading. There were limited data reported on the pool thermal behavior.

The specific problem of stratification in BWR suppression pools was addressed by Katakoa et al. in an experimental study of a water-wall suppression pool design\textsuperscript{[9]}. They observed very strong stratification above an electric heat source submerged in a water pool. Heat transfer into the volume below the heat source was accurately predicted by one-
dimensional transient conduction analysis, indicating that the lower volume participated only through conduction. However, main mechanism of pool mixing at WW is convection generated by inertia of high momentum steam and buoyancy force generated by density differences caused by the temperature distribution. Moreover, one-dimensional model is not enough to explain the three-dimensional pool circulation and convection.

The general problem of turbulent, transient natural convection induced by a shallow source of momentum and heat has received little focused attention. Analysis of the problem requires the synthesis of techniques developed for turbulent natural convection in enclosures and for turbulent buoyant jets. A number of researchers solved turbulent natural convection problems numerically using k-ε turbulence model. Farouk applied the k-ε turbulence model to the prediction of turbulent buoyant driven convective heat transfer with internal heat sources in a rectangular cavity\textsuperscript{[10]}. 

Buoyant jets were studied extensively, as summarized by Gebhart et al.\textsuperscript{[11]}. Tenner and Gebhart studied upward low-momentum laminar buoyant jets of fresh water into linearly stratified salt water\textsuperscript{[12]}. The buoyant jet induced the flow of a toroidal cell around itself, drawn up by the viscous shear of the jet.

Chen and Rodi reviewed experimental investigations of turbulent buoyant plumes\textsuperscript{[13]}. Transient phenomena that were extensively investigated include thermals, where an isolated burst of low density fluid rises through a stagnant fluid. Tuner studied transient plumes suddenly started from a source of buoyancy, showing that the advancing front possesses a cap-like structure similar to a thermal, while the following portion of the plume possesses a self-similar structure\textsuperscript{[14]}. Turner found that the transient plume permits a similarity solution, when the front is assumed to have a lower velocity than a pure thermal.
Peterson et al and Fox et al. performed tests of thermal stratification in the pool\cite{15,16} to simulate the Passive Containment Condenser System vent discharge into the suppression pool. When a heat and momentum is located close to the surface of a stagnant liquid pool, transient thermal stratification occurs upon activation of the source. Peterson et al presented a detailed experimental and numerical investigation of such two-dimensional transient stratification and Fox et al formulated one-dimensional control volume model using the assumption of perfect stratification. For the range of parameters studied here, the initial development of a buoyant jet and its spread across the pool surface generates a layer of light fluid on the pool surface. This layer grows in depth until it reaches the jet region. Strong stratification can occur when a buoyant plume is submerged at a shallow depth in an initially stagnant pool, both in the cases of laminar and turbulent jets. This stratification limits the volume of the pool available as a heat sink. The region below the source of momentum and heat remains inactive as a heat sink.

The densimetric Froude number is used to characterize the force balance within the buoyant plume. This is given by Eq. 1.1:

$$Fr = \frac{U}{\sqrt{g \frac{\rho_a - \rho_{\text{pl}}}{\rho_a} D_0}} = \frac{\text{momentum flux}}{\text{buoyancy force}}$$  \hspace{1cm} \text{Eq. 1.1}

Gamble et al. refer to the Richardson number that defines the case when the inertia of jet causes stratification to degrade in the SP\cite{5}, which is defined as in Eq. 1.2.
For pure steam injection into a water pool, bubble formation process is determined by the net steam mass accumulation in the bubble, which is the difference of the inlet mass and the condensed mass. Some experiments (Chan and Lee, Liang and Griffith) were performed under the atmosphere pressure\cite{17, 18}. With the limited observations, the formation process was roughly divided into the chugging, bubbling and jetting flow regimes based on the characteristics of the interface structures. Chan and Lee plotted their data in the coordinates of the steam mass flux and pool liquid temperature. It was found that the regime is mainly determined by steam mass flux. Liang and Griffith have given the transition criteria between these regimes based on energy balance and condensation mechanisms.

Theofanous et al. examined the problem of predicting mass transfer coefficients for gas absorption by turbulent liquids\cite{19}. Forced submerged vertical turbulent jet flows were considered as the primary mixing mechanism. Two approaches, based on idealized eddy structures of turbulence and the concept of surface-tension-damped laminar sub layer, were utilized to estimate mass transfer at a free surface. The jet Reynolds number determines whether the jet will be turbulent. This is given by Eq. 1.3.

\[
\text{Re}_\text{jet} = \frac{\rho U_0 D_0}{\mu} = \frac{\text{inertia force}}{\text{viscous force}}
\]  

Eq. 1.3
Gamble et al.\textsuperscript{[5]} defines a Froude number that governs the impingement interaction of the jet in terms of the jet thickness at the vertical location from the impingement stagnation point, given by Eq. 1.4.

\[
Fr_{imp} = \left( \frac{U^2}{g \left( \delta \right)} \right) \text{ Eq. 1.4}
\]

Current nuclear reactor system safety analysis codes, such as RELAP5, do not consider condensation caused by a submerged jet within the pool and at the surface. The McAdams natural convection correlation for stratified flow is utilized to model heat transfer between bulk liquid and a saturated interface\textsuperscript{[20]}. The correlation relates the Nusselt number and Rayleigh number as Eq. 1.5.

\[
Nu = 0.27 Ra^{0.25} \text{ Eq. 1.5}
\]

In the SP, the actual heat transfer is expected to be stronger than natural convection phenomena due to mixing and condensation introduced by a steam/noncondensable gas jet. Various experiments that address the condensation phenomena caused by turbulent mixing at a free-surface are available (Brown et al, Sonin et al)\textsuperscript{[21, 22]}. The efficiency of heat transfer in an enclosure also depends on the plume shape. Chun et al, Kim et al and Song et al defined three general idealized shapes of the pure steam jet plume in a subcooled pool\textsuperscript{[23, 24, 25]}. These are ideal conical, ellipsoidal and divergent shapes. The justification for defining these idealized shapes were based on experimental
observations, where the plume shape and length were found to depend on the injection diameter, injection orientation, pool subcooling and steam mass flux. A correlation is available (Chun et al) to predict the length to diameter ratio of the plume as a function of the condensation driving potential and steam mass flux. This is given by Eq. 1.6.

\[ h = 1.3583c_p G_m B^{0.0405} \left( \frac{G_0}{G_m} \right)^{0.3614} \]  

Eq. 1.6

This correlation excluded the mean vapor transport modules as this quantity cannot be directly measured in experiment\([23]\). However, Chan & Lee identified the transport transfer processes in the vapor and liquid regions near the interfaces as the source governing the complex behavior of the interface\([17]\). Their works models direct condensation of pure steam, but cannot explain direct condensation of steam and noncondensable gas mixture.

The modeling of stratification and mixing in a large enclosed volume, for computational purposes, needs to consider two parts that naturally arise: the fluid contained within the buoyant jet and the fluid in the ambient volume (Christensen & Peterson)\([26]\). The Lagrangian approach by Christensen & Peterson was adopted to eliminate ‘numerical diffusion.

Some unique phenomena were observed in the air-steam mixture experiments. Meier, Andreani, and Yadigaroglu found that most of the steam is condensed even before the bubble is detached from the nozzle exit. The remaining steam inside the bubble quickly reaches the thermal equilibrium with surrounding water. This process would be enhanced
by the bubble break up. They concluded that correlation or formula for the process may be difficult to be given explicitly.

1.4 Objectives and Scope of Study

Main objective of this thesis is to assess the safety systems design of the next generation boiling water reactor through the analysis of the experiment data produced by the integral test facility and through the calculation of the best estimation reactor system safety code, such as RELAP5. The global trends as well as important local phenomena accompanying the MSLB LOCA transient are examined in order to investigate the overall performance of the reactor safety systems. The scaling methodology used to design the integral test facility is evaluated through the comparison of the RELAP5 code calculation results with the experiment data. The distortions between the test facility and the ideal scaled facility are highlighted and their effects on the facility behavior are discussed.

One important local phenomenon, the suppression pool mixing and condensation, is further investigated through performing separated-effect test. Numerical calculation to simulate this problem is carried out to provide better understanding on this issue. Finally, a simple analytical model which can evaluate the pool mixing caused by the bubble plume and hot liquid plume will be developed. This model should be applicable to predict the suppression pool mixing process in both the PUMA facility and the SBWR. In a summary, the specific objectives of this research are to:

1. Identify important phenomena following a LOCA in the next generation boiling water reactor.
2. Analyze the main steam line break integral test data performed in the PUMA facility.

3. Assess the safety systems behavior of the next generation boiling water reactor by performing experiment and numerical simulation.

4. Evaluate the scaling methodology by comparing the calculation results of the best estimation reactor system safety code with the experiment data.

5. Investigate the important local phenomenon that affects reactor containment pressure.

6. Develop the analytical model to predict the suppression pool mixing process.

7. Compare the model predictions with PUMA separated-effect test data.
CHAPTER 2. MAIN STEAM LINE BREAK TEST IN THE PUMA FACILITY

2.1 Main Steam Line Break Accident in SBWR

The main steam line (MSL) break test in the PUMA facility\cite{27} simulates the large break loss of coolant accident (LB-LOCA) in SBWR. This LOCA transient is initiated by assuming that a double ended pipe break will take place at one of the main steam lines. The main steam line is a 28 inch pipe line that transfers the steam from reactor vessel to the steam turbine. After the MSL break initiated, steam will be discharged from RPV into the drywell through the broken main steam line. A flow restriction nozzle is installed on each MSL, close to the connection point to RPV. When the pressure difference between RPV and drywell is large, steam flow will be choked at the throat of the nozzle.

Figure 2.1\cite{2} shows three major phases after MSL break initiated, namely, the blowdown phase, the GDCS injection phase and the long-term cooling phase. The blowdown phase lasts for short time period. During this phase, the nuclear reactor will be automatically shut down once abnormal high pressure inside containment is detected. The nuclear fuel rods keeps releasing decay energy into the coolant. During this period, the RPV pressure quickly decreases from 7 MPa to about 350 kPa through 1). steam discharging into drywell through the break line; 2). steam discharging into drywell and suppression pool through the ADS lines. The ADS will be automatically activated once the low RPV water level signal (L1) is confirmed to last for at least 10 seconds.
Three squib valves on the GDCS drain lines will be opened at 150 seconds after the ADS is activated. However, GDCS water injection will not start at this moment due to the fact that GD water head cannot overcome the positive pressure difference between the RPV and the drywell. In this phase, the noncondensable gas initially filled inside the drywell will be pushed into the suppression pool through the DW/SP vertical vent lines. Due to the large pressure difference between the drywell and suppression pool, the PCCS works on the bypass mode. The steam/noncondensable mixture will be forced to flow from the drywell into suppression pool through PCCS vent lines.

The containment pressure keeps increasing as it keeps receiving large amount of steam discharged from RPV. When the water head in the gravity driven cooling system tank equals to the positive pressure difference between the RPV and drywell, plus the cracking pressure of the check valve installed in the GD drain lines, GDCS water starts to be injected into the RPV. This marks the initiation of the GD injection phase. Boiling inside RPV will be inhibited after the vessel receives the subcooled GD water. Steam flowing from RPV to drywell gradually stops. Drywell pressure starts to drop due to the steam condensation on containment wall and on the contacting surface to suppression pool water. On the other hand, since the suppression pool gas space is filled with noncondensable gas, suppression pool pressure cannot decrease too much during this period. When the drywell and the SP pressure difference drops below the cracking pressure (3.45 kPa) of the check valves installed in the vacuum breaker lines, the check valves open and noncondensable gas flows back from suppression pool to drywell. The check valves will close once the drywell pressure equals to the SP pressure. In this period, the PCCS does not work because 1). the driving pressure difference for steam flowing into PCCS disappeared; and 2).
noncondensable gas inside PCCS condenser tubes cannot be purged into the suppression pool. GD water injection keeps RPV inventory increasing until the RPV water level reaches the DPV line elevation. Then the injected GD water will overflow into the drywell through the DPV opennings. GD tank water elevation gradually drops to the same water elevation inside RPV.

The core decay heat eventually will heat up injected GD water to saturation temperature. RPV re-boiling happens and this marks the beginning of the long-term cooling phase. In this phase, RPV releases steam into containment through the broken main steam line and through ADS lines. Released steam will be pushed into PCCS condenser and will be condensed there. Core decay heat carried by the steam will then be transferred to the PCCS pool water. The condensate from PCCS condenser will first flow into the GDCS tank, then drains back into the reactor vessel. Noncondensable gas inside PCCS condenser tubes will be periodically purged into the suppression pool gas space by the pressure difference between the drywell and suppression pool.
2.2 Initial Conditions for the PUMA MSLB Test\textsuperscript{[27]}

The MSL break test of PUMA facility is carried out by following the exact accident procedure in the SBWR, except that PUMA only simulates the LOCA transient after RPV pressure drops below 1.03 MPa (150 psia). The reduction of the maximum pressure can simplify the test facility design and reduce the cost of the facility. The primary concern of the integral test facility is to reproduce the phenomena encountered after the reactor vessel
is depressurized and the GDCS is activated. Thus, such simplification will not affect the value of data collected in the integral test.

One technical problem generated by this simplification is that when the RPV pressure drops to 1.03 MPa in the blowdown phase, major components of the SBWR (drywell, SP, GD pool, ICS and PCCS pool) will be heated-up from the normal operation conditions by the discharged RPV steam. The thermodynamic status of all facility components at the test starting point can be predicted by the reactor system transient analysis code such as the RELAP5. The initialization preparation should be performed for PUMA test facility in order to reach the correct status. The detailed PUMA facility initialization procedure is explained in the Section 2.3.

The RELAP5 input deck for SBWR main steam line break is used to generate the PUMA test initial conditions. This RELAP5 input deck is built based on the SBWR standard safety analysis report. All major components, such as the RPV, DW, ICS, PCCS and GDCS, and the connection pipe lines between them are included in the input deck. The heat structures in the vessel and containment are also considered in the input deck. A steady state running at the normal operation condition is performed to ensure that the input deck has correct initial conditions indicated by the SBWR design report. The transient running for the MSLB is terminated once the RPV pressure reaches 150 psia. Table 2.1 lists the initial conditions for PUMA MSLB test. The pressure and temperature values come directly from the predicted values from the code calculation. The inventory values, such as the initial water level inside RPV, should be scaled down by four times from the code calculation results.
Table 2.1 Initial Conditions for PUMA MSLB Test

<table>
<thead>
<tr>
<th>Component</th>
<th>Parameter</th>
<th>Required Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>RPV</td>
<td>Steam Dome Pressure (kPa)</td>
<td>1034</td>
</tr>
<tr>
<td></td>
<td>Steam Dome Temperature (°C)</td>
<td>186</td>
</tr>
<tr>
<td></td>
<td>Collapsed Water Level (m)</td>
<td>2.64</td>
</tr>
<tr>
<td>Upper Drywell</td>
<td>Pressure (kPa)</td>
<td>235</td>
</tr>
<tr>
<td></td>
<td>Steam/NC Temperature (°C)</td>
<td>127</td>
</tr>
<tr>
<td>Lower Drywell</td>
<td>Steam/NC Temperature (°C)</td>
<td>107</td>
</tr>
<tr>
<td></td>
<td>Water Temperature (°C)</td>
<td>92</td>
</tr>
<tr>
<td></td>
<td>Water Level (m)</td>
<td>0.1167</td>
</tr>
<tr>
<td>Suppression Pool</td>
<td>Pressure (kPa)</td>
<td>231</td>
</tr>
<tr>
<td></td>
<td>Gas Space Temperature (°C)</td>
<td>63</td>
</tr>
<tr>
<td></td>
<td>Water Temperature (bulk) (°C)</td>
<td>58</td>
</tr>
<tr>
<td></td>
<td>Water Level (m)</td>
<td>1.69</td>
</tr>
<tr>
<td>GDCS</td>
<td>Water Temperature (°C)</td>
<td>58</td>
</tr>
<tr>
<td></td>
<td>Water Level (m)</td>
<td>1.25</td>
</tr>
<tr>
<td>ICS</td>
<td>Steam Temperature (°C)</td>
<td>186</td>
</tr>
<tr>
<td></td>
<td>Pool Temperature (°C)</td>
<td>43</td>
</tr>
<tr>
<td></td>
<td>Water Level (m)</td>
<td>1.1</td>
</tr>
<tr>
<td>PCCS</td>
<td>Steam Temperature (°C)</td>
<td>127</td>
</tr>
<tr>
<td></td>
<td>Pool Temperature (°C)</td>
<td>38</td>
</tr>
<tr>
<td></td>
<td>Water Level (m)</td>
<td>1.1</td>
</tr>
</tbody>
</table>
2.3 Initialization Procedure for the PUMA MSLB Test

As that has been explained in the previous section, the PUMA initialization procedure should be performed before the integral test in order to heat up the facility to the desired hot status predicted by the RELAP5 code. The normal operation conditions for the containment are listed in the SBWR SSAR. The PUMA MSLB initialization procedure utilizes continuous steam blowdown from the RPV to bring the containment to the desired hot status. In order to control the time period for steam blowdown, it is essential to estimate the energy that the containment will receive when the RPV pressure drops from normal operation pressure to 150 psia.

Total blowdown energy to the containment consists of the energy stored in the steam and water in the containment, and the energy stored inside the containment wall. It is difficult to estimate these two parts by using theoretical model. Here the RELAP5 calculation results for SBWR MSLB are analyzed in order to give the value of total blowdown energy. The energy stored in the steam and water is directly given by the code results of the total internal energy inside containment. The wall stored energy is calculated by integrating the code predicted wall heat flux. Finally, total blowdown energy to the containment is estimated to be 2.6941E+9 J in the SBWR. Total blowdown energy to the containment in the PUMA is calculated by scaling down the SBWR value 400 times, which will be 6.735E+8 J. The time period required for steam blowdown can be calculated by dividing total blowdown energy with the heater power. When the RPV pressure is set to be 120 psig and the heater power is set to be 250 kW, the time period is calculated to be 2700 seconds.
Results of initialization process for MSLB test are shown in Table 2.2. Only parameter that has obvious discrepancy from the desired initial value is the lower DW water temperature (measured by TE-DW-08). This temperature discrepancy happens due to the fact that by design, the PUMA facility has more heat loss in the lower DW than the SBWR.

The overall energy balance for the condensate in the lower drywell will be:

$$h \times A_{\text{surface}} \times (T_w - T_\infty) = m_w \times C_p \times \frac{dT_w}{dt}$$  \hspace{1cm} \text{Eq. 2.1}

Here, $h$ is the heat transfer coefficient [W/(m$^2$.K)], $A_{\text{surface}}$ is the surface area of the lower DW water region [m$^2$], $T_w$ is the averaged lower DW water temperature [K], $T_\infty$ is the environment temperature [K], $m_w$ is the total mass of the lower DW water [kg], $C_p$ is the heat capacity of the lower DW water [J/(kg.K)], and $t$ is time [s].

Lower DW water mass can be calculated from density and volume:

$$m_w = \rho \times V_w$$  \hspace{1cm} \text{Eq. 2.2}

Thus Eq. 2.1 can be rearranged to:

$$\frac{dT_w}{dt} = \frac{h \times A_w}{\rho \cdot C_p \cdot V_w} \cdot (T_w - T_\infty)$$  \hspace{1cm} \text{Eq. 2.3}

Assuming that all properties are well scaled down from PUMA to SBWR,
\[
\frac{dT_w}{dt} = \left(\frac{A_w}{R}\right)_R = \left(\frac{\pi DL}{D^2 L}\right)_R = \left(\frac{1}{D}\right)_R = \frac{1}{10}
\]

Eq. 2.4

Eq. 2.4 shows that by design, the water temperature decreasing rate in the PUMA facility lower drywell will be much higher than that in the SBWR.

Table 2.2 PUMA Facility Status after the Initialization Process for MSLB Test

<table>
<thead>
<tr>
<th></th>
<th>Required Value</th>
<th>Measured Value</th>
<th>Relative Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Upper DW Pressure (kPa)</td>
<td>235</td>
<td>225.4</td>
<td>-4.09</td>
</tr>
<tr>
<td>Upper DW Temperature ((^\circ)C)</td>
<td>127</td>
<td>131.0</td>
<td>3.15</td>
</tr>
<tr>
<td>Lower DW Temperature ((^\circ)C)</td>
<td>92</td>
<td>44.8</td>
<td>-51.3</td>
</tr>
<tr>
<td>Lower DW Water Level (m)</td>
<td>0.1167</td>
<td>0.1063</td>
<td>-8.91</td>
</tr>
<tr>
<td>SP Pressure (kPa)</td>
<td>231</td>
<td>224.68</td>
<td>-2.74</td>
</tr>
<tr>
<td>SP Gas Space Temperature ((^\circ)C)</td>
<td>63</td>
<td>63.0</td>
<td>0</td>
</tr>
<tr>
<td>Averaged SP Water Temperature ((^\circ)C)</td>
<td>58</td>
<td>57.94</td>
<td>-0.1</td>
</tr>
<tr>
<td>SP Water Level (m)</td>
<td>1.69</td>
<td>1.657</td>
<td>-1.95</td>
</tr>
<tr>
<td>RPV Water Level (m)</td>
<td>2.64</td>
<td>2.62</td>
<td>-0.76</td>
</tr>
</tbody>
</table>

2.4 Decay Power for the PUMA MSLB Test

Ideally the PUMA heater power should be scaled down from the SBWR decay power table, both in time and in the power value. The recommended decay power curve for the SBWR is proposed in reference\(^2\). The scaled down decay power curve for PUMA is listed
in Table 2.3. Here, time zero corresponds to the reactor SCRAM moment, which happens when RPV collapsed water reaches Level 3 (17.333 m in SBWR).

<table>
<thead>
<tr>
<th>Time (second)</th>
<th>Decay Heat Fraction</th>
<th>Decay Heat Power (kW)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>1.0</td>
<td>10000</td>
</tr>
<tr>
<td>0.05</td>
<td>0.98281</td>
<td>98281</td>
</tr>
<tr>
<td>0.5</td>
<td>0.33403</td>
<td>33403</td>
</tr>
<tr>
<td>1.0</td>
<td>0.15113</td>
<td>15113</td>
</tr>
<tr>
<td>2.0</td>
<td>0.07043</td>
<td>7043</td>
</tr>
<tr>
<td>3.0</td>
<td>0.0578</td>
<td>578</td>
</tr>
<tr>
<td>4.0</td>
<td>0.05368</td>
<td>536.8</td>
</tr>
<tr>
<td>5.0</td>
<td>0.04964</td>
<td>496.4</td>
</tr>
<tr>
<td>30.0</td>
<td>0.0467</td>
<td>467</td>
</tr>
<tr>
<td>100.0</td>
<td>0.0358</td>
<td>358</td>
</tr>
<tr>
<td>200.0</td>
<td>0.0309</td>
<td>309</td>
</tr>
<tr>
<td>500.0</td>
<td>0.0245</td>
<td>245</td>
</tr>
<tr>
<td>1000.0</td>
<td>0.0192</td>
<td>192</td>
</tr>
<tr>
<td>4000.0</td>
<td>0.013</td>
<td>130</td>
</tr>
<tr>
<td>5000.0</td>
<td>0.012</td>
<td>120</td>
</tr>
<tr>
<td>20000.0</td>
<td>0.00812</td>
<td>81.2</td>
</tr>
<tr>
<td>40000.0</td>
<td>0.00664</td>
<td>66.4</td>
</tr>
<tr>
<td>50000.0</td>
<td>0.00624</td>
<td>62.4</td>
</tr>
</tbody>
</table>
The PUMA experiment is designed to start at the blowdown condition of 1030 kPa. The SBWR MSLB code predicts that the RPV pressure drops to 1030 kPa at 220 seconds after the break is initiated. At that moment, the decay power for PUMA is 353 kW.

The PUMA experiment heater power should be modified to compensate the distortions of stored energy released from RPV internal structures, especially the stored energy released from the fuel rods. This distortion is again caused by the PUMA experiment starting from 1030 kPa. In the real depressurization process, the stored energy inside fuel rods needs time to be transferred into the coolant. This process can be modeled by the one-dimensional transient heat conduction equation if uniform heat generation rate inside fuel rod and uniform heat convection boundary condition are assumed. Figure 2.2 shows the approximate temperature profile inside fuel rod at 1030 kPa and at the long term of the MSLB LOCA. The difference between the volume averaged temperature times heat capacity gives the value of stored energy that will be released into the coolant. The scaled down stored energy should be added into the PUMA decay power curve.

![Figure 2.2 Fuel Rod Temperature Profile at 1030 kPa and at Long Term](image)
2.5 General Discussion of the PUMA MSLB Test

Measured pressure in RPV, DW and SP during the PUMA MSLB test is shown in Figure 2.3. After the break is initiated, the RPV pressure drops from 1 MPa to about 260 kPa in 250 seconds. GDCS injection starts at 250 second when containment pressure rises close to the RPV pressure. During the GDCS injection phase, pressure in all components keeps decreasing due to RPV boiling is stopped. RPV re-boiling happens around 798 second. Pressure starts increasing from the lowest value (around 189 kPa) to 255 kPa. In the final stage of the long term cooling phase, the system pressure stabilized at around 255 kPa. The enlarged pressure trend in Figure 2.4 shows that SP pressure is higher than DW pressure during the GDCS injection phase.

![Figure 2.3 Pressure Trend during the PUMA MSLB Test](image-url)
Figure 2.4 Enlarged Pressure Trend during the PUMA MSLB Test

Measured collapsed water level inside RPV, DW and GD during the PUMA MSLB test is shown in Figure 2.5. Elevation zero in the figure refers to bottom of RPV. During the blowdown phase, RPV keeps losing water due to the steam discharging from the break line and from the ADS. RPV water level decreases to the minimal value before the GD injection starts. The minimum collapsed water level inside RPV is still higher than the top of active fuel. During the GDCS injection phase, the RPV collapsed water level keeps increasing until it reaches the DPV line penetration elevation; then water injected from GD tank directly flows into the DW through the DPV lines. The water level in the DW increases fast due to GD injection water overflow. The GD injection flow stops when the water level inside GD tank drops to the DPV line penetration elevation. In the long-term cooling phase, RPV and GD water slightly decreases due to the steam condensation inside containment. Figure 2.6 shows the measured GDCS A injection flow rate during the PUMA MSLB test.
Figure 2.5 RPV, DW and GD Water Level during the PUMA MSLB Test

Figure 2.6 GDCS A Injection Flow Rate during the PUMA MSLB Test
Measured ADS actuation sequences are shown in Figure 2.7. The ADS starts at 5 seconds after RPV collapsed water level drops to and maintains below Level 1, which is defined as 2.606 m in the PUMA facility. Measured RPV collapsed water level reaches 2.606 m around 163 second after the break initiated. Thus the first ADS valve (SRV-B) is activated around 168 second. After that, the SRV-B, the DPV on the unbroken MSL, the DPV-A and finally the DPV-B opens according to the pre-set time intervals.

The ICS and PCCS performance are shown in Figure 2.8. Test data indicates that both ICS and PCCS work well from the beginning time. However, the core decay heat power is much larger than the total heat removal capability of ICS and PCCS. Part of the un-carried away core decay energy is used to heat up the injected GDCS water. The remaining core decay energy is discharged into the SP through the horizontal vent openings.

After the GD injection happens, the DW pressure drops and the noncondensable gas is released back from SP to DW through the vacuum breaker lines. The noncondensable gas that enters the ICS condenser will stay there and eventually prevents steam condensation inside the ICS condenser. Thus, the ICS is not functional in the long-term cooling phase. However, the PCCS still works well in the long-term cooling phase because the noncondensable gas inside PCCS condenser can be purged into the SP through the PCCS vent lines. Test data shows that the core decay energy is mainly removed by the PCCS in the long-term cooling phase.
Figure 2.7 ADS Actuation Sequences during the PUMA MSLB Test

Figure 2.8 ICS and PCCS Performance during the PUMA MSLB Test
CHAPTER 3. RELAP5 SIMULATION FOR THE MSLB LOCA

3.1 Evaluating MSLB by Utilizing RELAP5 Code

The RELAP5 simulation for SBWR MSLB is essential to provide the initial conditions for performing PUMA integral test. RELAP5 is a well-known system code suitable for the analysis of all transients and postulated accidents in Light Water Reactor (LWR) systems, including both large and small break LOCAs as well as the full range of operational transients[28]. The one dimensional RELAP5 code is constructed from 6-equation two-fluid model for gas and liquid phases, and is solved by a fast, partially implicit numerical scheme to permit economical calculation of system transients. RELAP5 is developed by the Idaho National Engineering Laboratory under the sponsorship of the U.S. Nuclear Regulatory Commission. The RELAP5/MOD3 code has been upgraded for application to SBWR. Various systems inside SBWR are represented by basic building blocks of the code such as hydraulic volumes, pipes, branches, junctions, time-dependent volumes and time-dependent junctions, and heat structures etc. The code has the capability to track noncondensible gases which are assumed to be in mechanical and thermal equilibrium with the steam. The RELAP5 input deck developed by the Brookhaven National Laboratory for SBWR MSLB application has been used. The initial conditions of all components come from the normal operation conditions of SBWR.
PUMA test facility is a scaled down integral test facility from the SBWR geometry. In reality, there is no real SBWR plant has ever been built and operated. Thus, it is impossible to compare the test data generated by PUMA facility to any real SBWR transient data. Therefore, the comparison between the scaled-up PUMA experiment data with the RELAP5 simulation results for SBWR is useful to confirm the scaling approach used in the design of the PUMA facility.

RELAP5 calculation for MSLB has also been performed at the PUMA facility level. Here, RELAP5 code can be validated by comparing the code prediction results with measured test data. In this test facility level calculation.

3.2 RELAP5 Simulation for the PUMA MSLB Test

The RELAP5 simulation for PUMA MSLB is necessary for checking code capability for SBWR applications. The PUMA MSLB RELAP5 input deck includes 1). hydraulic component part that simulates all PUMA vessels and the connection pipe lines between them, 2). heater structure part that simulates the heater rods, ICS/PCCS condenser tubes and general wall heat structures inside RPV and containment, and 3). control variables and trip system that sets the boundary conditions of the transient, such as the break initiation and the ADS actuation sequences in the LOCA\[27\].

The RPV is the most complicated and crucial component in the facility. The lower plenum and the core inlet plate are modeled as branches component based on the suggestions from the RELAP5 user guide. Four parallel channels with cross-flow junctions are used to model the inner heater ring, middle heater ring, outer heater ring and the core bypass flow channel. The core flow channels are attached with heater structures which can
simulate the heat generation rate of heater rods in experiment. All other sections, such as the chimney, the separator, the dryer and the downcomer are modeled by pipe component.

The modeling of DW and SP is simple because essentially they are just two large tanks without energy source. The upper DW connects to the break line and various ADS pipe lines, thus it is modeled by the branch component. The vertical vent pipe and the SP are modeled as two parallel pipes that are with three cross-flow junctions. The three cross-flow junctions simulate the horizontal vent openings.

The modeling of GDCS follows the real facility geometry. Three GDCS tanks, the cover gas lines to the DW and the drain lines to the RPV are modeled by the pipe component.

The modeling of ICS and PCCS are challenging because it is difficult to simulate the condensation process inside the heat exchanger tubes and the PCCS venting process. A simple heat transfer coefficient look-up table is utilized in the input deck for modeling the ICS/PCCS condenser. Unlike in the experiment, the ICS/PCCS pool is not modeled in the RELAP5 input deck. An infinite large pool filled with 100 °C water is assumed in the code calculation.

The RELAP5 predicted RPV pressure is compared with test data in Figure 3.1. Overall agreement is good while in the blowdown phase, the code over-predicts the test results. This discrepancy comes from the slightly difference on the break flow rate, which is shown in Figure 3.2.

The comparison between the RELAP5 predicted RPV collapsed water level with the test data is shown in Figure 3.3. The code calculation has a delayed GDCS injection starting time. Therefore, the code predicts lower RPV water level in the GDCS injection
phase. The discrepancy disappears after the water level reaches the DPV penetration elevation. In the long term cooling mode, the code predicts lower RPV collapsed water level.

The lower DW water level trend comparison is shown in Figure 3.4. The water level initially increases slightly due to the steam condensation on the DW wall. After the RPV water level reaches the DPV penetration elevation, the DW water level increases quickly due to the RPV water overflow. Here again due to the delay of the predicted GDCS injection starting time, the DW water level in code stays below the test data in the RPV overflow phase.

![Figure 3.1 Comparison for RPV Pressure](image-url)
The GDCS A injection flow rate comparison is shown in Figure 3.5. The overall prediction of the code follows closely with the test data. Numerical oscillation is observed after the GDCS draining head drops to the value comparable to the cracking pressure of the check valve installed in the GDCS drain lines.

Figure 3.6 shows that the RELAP5 over predicts the DW pressure both in whole LOCA transient. The DW pressure is closed related to the condensation and mixing process happened inside suppression pool. It is suspected that the 1-D RELAP5 code lacks the capability to accurately model the condensation and mixing process inside SP. A check of the SP water temperature predicted by the code calculation proves that mixing is not modeled correctly inside the SP.
The overall ICS and PCCS performance predicted by the RELAP5 is shown in Figure 3.7. The ICS and PCCS total heat removal capability is close to the test data. In the code prediction the ICS still contribute a little in the decay heat removal.

The comparison between the PUMA RELAP5 code prediction and the experiment data indicated that the RELAP5 code can give reasonable predictions on the overall thermodynamic status of the facility during the MSLB LOCA transient. The functions of GDCS, ICS and PCCS in the LOCA have been correctly simulated. Thus, in general RELAP5 can be used in the SBWR applications. However, special attentions should be paid on simulating some local phenomena, such as the blowdown flow rate and the suppression pool condensation and mixing.

![Figure 3.3 Comparison for RPV Collapsed Water Level](image-url)
Figure 3.4 Comparison for Lower DW Water Level

Figure 3.5 Comparison for GDCS A Injection Flow Rate
Figure 3.6 Comparison for DW Pressure

Figure 3.7 RELAP5 Predicted ICS and PCCS Performance
3.3 Scaled-up PUMA Test Data Compared with SBWR Simulation

A comparison between the scaled-up PUMA test data with the SBWR RELAP5 simulation will validate the PUMA scaling approach. In the following figures, the PUMA data will be scaled-up in time by 2 times, in water level by 4 times and in mass flow rate by 200 times. The scaled-up PUMA data will be shifted 220 seconds because this is the time when the RPV pressure reaches 1030 kPa.

Figure 3.8 shows the comparison of the RPV pressure between the scaled-up test data and the code prediction. The general trend fit well except that the code has a faster depressurization rate. The long term pressure difference between them is close to 30 kPa, which may be caused by the containment pressure difference in the long term. Figure 3.9 shows the comparison of the DW pressure between the scale-up test data and the code prediction. Except for the value difference in the long term phase, another important difference is also shown in this figure. The test data of DW pressure is unchanged in the late portion of the test, which indicates that the decay heat power released by the heater rods is balanced by the ICS and PCCS heater removal capability in that period. However, the code predicted DW pressure still increasing at the end of the long term cooling phase.

The RPV collapsed water level comparison is shown in Figure 3.10. This figure shows that the minimum RPV water level in the code is lower than the scaled-up test data, which is caused by the late GD injection in the code. Figure 3.11 shows the comparison of GDCS loop A injection flow rate between the scaled-up test data and the code prediction for SBWR MSLB. However, both code prediction and experiment data show that the top of active fuel will always be covered by the coolant, thus, the fuel rods are safe in the whole transient. The RPV water level keeps same after the GD injection phase terminated.
because then the water level is same as the DPV line penetration elevation. The good agreement between the RELAP5 SBWR prediction and scaled-up PUMA test data demonstrates the scaling approach used to design and built PUMA facility is successful.

Figure 3.8 Comparison of RPV Pressure in SBWR MSLB
Figure 3.9 Comparison of DW Pressure in SBWR MSLB

Figure 3.10 Comparison of RPV Collapsed Water Level in SBWR MSLB
Figure 3.11 Comparison of GDCS A Injection Flow Rate in SBWR MSLB
CHAPTER 4. SUPPRESSION POOL CONDENSATION AND MIXING TESTS

4.1 Separated-effect Test on SP Condensation and Mixing

4.1.1 SP Separated-effect Test Facility\textsuperscript{[29]}

The PUMA facility was utilized to perform the SP condensation and mixing test. The PUMA facility is an integral test facility that is carefully scaled down from the SBWR design. It can simulate the whole LOCA event after the RPV is depressurized below 1.03 MPa (150 psi)\textsuperscript{[3]}.

The design of the PUMA facility was based on the three level scaling methods. The first level of scaling is based on the well-established approach obtained from the integral response function, namely, the integral scaling. This level insures that the steady-state as well as dynamic characteristics of the loops are scaled properly. The second level scaling is for boundary flow of mass and energy between components. This insures that the flow and inventory are scaled correctly. The third level of scaling is focused on the key local phenomena and constitutive relations. The facility has 1/4 height and 1/100 area ratio scaling. This corresponds to the volume scale of 1/400. The power scaling is 1/200 based on the integral scaling. The time will be twice faster in the model as predicted by the present scaling method. The scaling ratios of the PUMA facility to SBWR-600 are summarized in Table 1.2.
Only RPV, DW, and SP were involved in the SP separated effect test, the PCCS, ICS, and GDCS were isolated from the test section. A schematic figure of the test facility is shown in Figure 4.1.

In the SP separated-effect test, the RPV was used to supply the steam. The RPV was filled with water to a level of 4.8 m at the beginning of the experiment, which was close to the elevation of DPV line penetration. The heater power was cross-calibrated by the vortex flow meter in order to obtain the accurate steam flow rate from RPV. The PUMA RPV has been equipped with 70 thermocouples, various P cells and DP cells to measure the temperature, pressure and collapsed water level.

The DW was used as an intermediate space in the SP separated-effect test. The PCCS steam supply lines were closed during the test so that DW was only connected with SP through the horizontal vent line. The steam generated from the RPV will first be discharged into the DW through the DPV lines or MSL, and then it will be pushed into the SP by the pressure difference between the DW and SP.

DW was initially filled with air (as a simulator of nitrogen in the SBWR containment). The DW air should be purged into the WW during the test initialization process (refer to Section 2.5). Thus during the test the DW is kept as approximately filled with pure steam.

The SP is connected to DW through the vertical vent system, which is comprised of eight vertical/horizontal vent modules. Each module consists of a vertical flow channel extending into the SP water with three horizontal vent pipes opening in the pool. In the event of a LOCA, the increased pressure inside the DW forces a mixture of steam, water and noncondensable gas to discharge through the DW/SP vent system. The steam quickly condenses inside the pool. The noncondensable gas rise and will be collected in the gas
space volume of the SP. For the SP separated-effect test, three horizontal vents were opened as shown Figure 4.2. The size of vent opening is 175 mm × 22 mm, which was determined by height and area scaling ratio from PUMA to SBWR. Vent submerge effect was also tested by changing the vent opening depth. Figure 4.2 shows the second level vent opening configuration.

A global valve has been installed at the top of the SP. During the test, the SP pressure can be maintained as approximately steady state by controlling the valve opening size. Thus the DW pressure can also be maintained as steady state.

Noncondensable gas concentration in the air-steam mixture discharging flow is an important test parameter. It is difficult to maintain a constant noncondensable gas concentration rate in the injection flow if air is mixed with steam inside the DW. For separated-effect test all air inside DW have been purged into WW during the initialization process, and external air was supplied and mixed with steam inside the vertical vent pipe. Before injection, air was preheated to a slightly higher temperature than the temperature of the steam from DW, so that steam condensation due to temperature difference between the air and steam was prevented. An air injection sparger was designed to mix air and steam uniformly. Two air mass flow controllers were installed to set the flow rate of the noncondensable gas for the test.
Figure 4.1 SP Separated-effect Test Facility

Figure 4.2 Vent Opening at the Second Level
4.1.2 Instrumentation\textsuperscript{[29]}

PUMA instrumentation consists of numerous devices that provide a detailed measurement of the temperature, pressure, collapsed water level and flow rate inside each component and connecting pipe lines. The instrumentation for the experimental facility is summarized in Table 4.1.

PUMA SP originally has 14 thermocouples installed to measure the pool water temperature and the gas space temperature. However, these measurements cannot give detailed information about thermal stratification in the SP pool. Therefore, 74 T-type thermocouples (Omega Engineering, Inc., Stanford, CT) were put in additionally for the SP separated-effect test. The positions of thermocouples were determined by predicting the jet flow direction from the horizontal vent. Two cages made of the thin stainless steel tubes (1/4” tube) were used to fix the thermocouples. Figure 4.3 shows the new added thermocouples of the vent direction inside the SP. Here in order to distinguish the position of thermocouples, ‘Vent Direction’ and ‘Pool Direction’ was defined. Thermocouples of ‘Vent Direction’ are installed facing the steam and air mixture jet flow coming from the horizontal vent, and thermocouples of ‘Pool Direction’ are installed on the opposite side of the pool.

Air mass controllers are installed to indicate flow rates and to set flow rate of the noncondensable. The specifications of the two air mass flow controllers are shown in Table 4.2.

A high speed camera was used to capture jet interfacial area structure during the direct condensation for short time. A digital camcorder was also used to record the jet surface
structure during the whole test. An underwater lamp was installed to provide the illumination.

All involved instrumentation has been calibrated before the test. The differential pressure gauge and the absolute pressure gauge have been calibrated by using the digital pressure calibrator (Druck DPI 601). In the experiment, the steam mass flow rate was controlled by manually setting the RPV heater power. Under the given DW pressure, this RPV heater power can be converted to the steam mass flow rate if the saturation condition and no heat loss are assumed. Figure 4.4 shows the calibration result for RPV heater controller by measuring the DPV line steam mass flow rate when the RPV heater power is 150 kW. The air line vortex flow meter has been calibrated by comparing the reading from air mass flow controller. Figure 4.5 shows the calibration results for the air line vortex flow meter.

Table 4.1 Instrumentation Used in the SP Separated-effect Test

<table>
<thead>
<tr>
<th>Component</th>
<th>Measured Parameter</th>
<th>Instrumentation</th>
</tr>
</thead>
<tbody>
<tr>
<td>RPV</td>
<td>Power</td>
<td>Heat controller</td>
</tr>
<tr>
<td></td>
<td>Temperature</td>
<td>Type K thermocouple</td>
</tr>
<tr>
<td></td>
<td>Pressure</td>
<td>Absolute pressure transducer</td>
</tr>
<tr>
<td></td>
<td>Water Level</td>
<td>Differential pressure transducer</td>
</tr>
<tr>
<td>Steam Supply Line</td>
<td>Flow Rate</td>
<td>Nozzle flow meter</td>
</tr>
<tr>
<td>(MS-A, MS-B, DPV-A,</td>
<td></td>
<td></td>
</tr>
<tr>
<td>DPV-B)</td>
<td>Flow Rate</td>
<td>Vortex flow meter</td>
</tr>
<tr>
<td></td>
<td>Temperature</td>
<td>Type K thermocouple</td>
</tr>
<tr>
<td></td>
<td>Pressure</td>
<td>Absolute pressure transducer</td>
</tr>
</tbody>
</table>
Table 4.1 Continued

<table>
<thead>
<tr>
<th>Air Supply Line</th>
<th>Mass Flow Rate</th>
<th>Air mass flow controller</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Volume Flow Rate</td>
<td>Vortex flow meter</td>
</tr>
<tr>
<td></td>
<td>Temperature</td>
<td>Type K thermocouple</td>
</tr>
<tr>
<td></td>
<td>Pressure</td>
<td>Absolute pressure transducer</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Drywell</th>
<th>Water Level</th>
<th>Differential pressure transducer</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Temperature</td>
<td>Type K thermocouple</td>
</tr>
<tr>
<td></td>
<td>Pressure</td>
<td>Absolute pressure transducer</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Suppression Pool</th>
<th>Temperature</th>
<th>Type K/T thermocouple</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Pressure</td>
<td>Absolute pressure transducer</td>
</tr>
<tr>
<td></td>
<td>Water Level</td>
<td>Differential pressure transducer</td>
</tr>
<tr>
<td></td>
<td>Visualization</td>
<td>High speed camera</td>
</tr>
</tbody>
</table>

Table 4.2 Air Mass Flow Controller Specifications

<table>
<thead>
<tr>
<th>Model</th>
<th>Volume Flow Rate (L/min)</th>
<th>Mass Flow Rate (g/sec)</th>
<th>Error Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>AALBORG GFC47</td>
<td>0 - 100</td>
<td>0 - 1.96</td>
<td>1.5 %</td>
</tr>
<tr>
<td>AALBORG GFC67</td>
<td>0 - 500</td>
<td>0 - 9.80</td>
<td>1.5 %</td>
</tr>
</tbody>
</table>
Figure 4.3 Vent Side Thermocouple Locations

Figure 4.4 Calculated Heater Power from Measured Steam Flow Rate
4.1.3 SP Separated-effect Test Matrix\cite{29}

The SP separated-effect test matrix has been obtained by using RELAP5 simulation for the blowdown phase of the SBWR-600. The main steam line break and bottom drain line break simulation results were analyzed. Figure 4.6 shows the key code prediction values. Values obtained from the RELAP5 simulation were scaled down according to the PUMA to SBWR scaling ratios. Thus for the PUMA geometry, the DW pressure range predicted by RELAP5 is from 204.7 kPa to 261.9 kPa, the steam mass flow rate range predicted by RELAP5 is from 0.007 kg/s to 0.129 kg/s, and the noncondensable gas concentration range in the vertical vent pipe predicted by RELAP5 is from 0.7% to 64.8%.
Based on these scaling down parameters and the PUMA facility capabilities, the test matrix for the SP separated-effect tests was prepared with valuable suggestions from the NRC staff. Thirty-two tests were performed, with the DW pressure respectively at 200 kPa, 230 kPa and 260 kPa, with the steam flow rate at 70 g/s and 120 g/s, with the SP initial water temperature at 40 °C, 50 °C and 60 °C, with the air mass concentration at 0, 0.5%, 2.5% and 5%. Among the 32 tests, 24 tests were performed with the vent opening at the first level (top vent), and 8 tests were performed with the vent opening at the second level (middle vent). Table 4.3 and Table 4.4 show the SP separated-effect test matrix.

### Table 4.3 Test Matrix for the Vent Opening at the 2nd Level

<table>
<thead>
<tr>
<th>Test No.</th>
<th>Drywell Pressure (kPa)</th>
<th>Steam Injection Rate (g/s)</th>
<th>Air Concentration (%)</th>
<th>SP Initial Temperature (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>SLEV1</td>
<td>230</td>
<td>70</td>
<td>0</td>
<td>60</td>
</tr>
<tr>
<td>SLEV2</td>
<td>230</td>
<td>70</td>
<td>0.5</td>
<td>60</td>
</tr>
<tr>
<td>SLEV3</td>
<td>230</td>
<td>70</td>
<td>2.5</td>
<td>60</td>
</tr>
<tr>
<td>SLEV4</td>
<td>230</td>
<td>70</td>
<td>5</td>
<td>60</td>
</tr>
<tr>
<td>SLEV5</td>
<td>230</td>
<td>120</td>
<td>0</td>
<td>40</td>
</tr>
<tr>
<td>SLEV6</td>
<td>230</td>
<td>120</td>
<td>0.5</td>
<td>40</td>
</tr>
<tr>
<td>SLEV7</td>
<td>230</td>
<td>120</td>
<td>2.5</td>
<td>40</td>
</tr>
<tr>
<td>SLEV8</td>
<td>230</td>
<td>120</td>
<td>5</td>
<td>40</td>
</tr>
</tbody>
</table>
Table 4.4 Test Matrix for the Vent Opening at the 2\textsuperscript{nd} Level

<table>
<thead>
<tr>
<th>Test No.</th>
<th>Drywell Pressure (kPa)</th>
<th>Steam Injection Rate (g/s)</th>
<th>Air Concentration (%)</th>
<th>SP Initial Temperature (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>LSF1</td>
<td>230</td>
<td>70</td>
<td>0</td>
<td>50</td>
</tr>
<tr>
<td>LSF2</td>
<td>230</td>
<td>70</td>
<td>0.5</td>
<td>50</td>
</tr>
<tr>
<td>LSF3</td>
<td>230</td>
<td>70</td>
<td>2.5</td>
<td>50</td>
</tr>
<tr>
<td>LSF4</td>
<td>230</td>
<td>70</td>
<td>5</td>
<td>50</td>
</tr>
<tr>
<td>LSF5</td>
<td>230</td>
<td>70</td>
<td>0</td>
<td>60</td>
</tr>
<tr>
<td>LSF6</td>
<td>230</td>
<td>70</td>
<td>0.5</td>
<td>60</td>
</tr>
<tr>
<td>LSF7</td>
<td>230</td>
<td>70</td>
<td>2.5</td>
<td>60</td>
</tr>
<tr>
<td>LSF8</td>
<td>230</td>
<td>70</td>
<td>5</td>
<td>60</td>
</tr>
<tr>
<td>LSF9</td>
<td>200</td>
<td>70</td>
<td>0.5</td>
<td>60</td>
</tr>
<tr>
<td>LSF10</td>
<td>260</td>
<td>70</td>
<td>0.5</td>
<td>60</td>
</tr>
<tr>
<td>HSF1</td>
<td>230</td>
<td>120</td>
<td>0.5</td>
<td>40</td>
</tr>
<tr>
<td>HSF2</td>
<td>230</td>
<td>120</td>
<td>2.5</td>
<td>40</td>
</tr>
<tr>
<td>HSF3</td>
<td>230</td>
<td>120</td>
<td>5</td>
<td>40</td>
</tr>
<tr>
<td>HSF4</td>
<td>230</td>
<td>120</td>
<td>0</td>
<td>50</td>
</tr>
<tr>
<td>HSF5</td>
<td>230</td>
<td>120</td>
<td>0.5</td>
<td>50</td>
</tr>
<tr>
<td>HSF6</td>
<td>230</td>
<td>120</td>
<td>2.5</td>
<td>50</td>
</tr>
<tr>
<td>HSF7</td>
<td>230</td>
<td>120</td>
<td>5</td>
<td>50</td>
</tr>
<tr>
<td>HSF8</td>
<td>230</td>
<td>120</td>
<td>0</td>
<td>60</td>
</tr>
<tr>
<td>HSF9</td>
<td>230</td>
<td>120</td>
<td>0.5</td>
<td>60</td>
</tr>
</tbody>
</table>
Table 4.4 Continued

<table>
<thead>
<tr>
<th>HSF10</th>
<th>230</th>
<th>120</th>
<th>2.5</th>
<th>60</th>
</tr>
</thead>
<tbody>
<tr>
<td>HSF11</td>
<td>230</td>
<td>120</td>
<td>5</td>
<td>60</td>
</tr>
<tr>
<td>HSF12</td>
<td>200</td>
<td>120</td>
<td>0.5</td>
<td>40</td>
</tr>
<tr>
<td>HSF13</td>
<td>260</td>
<td>120</td>
<td>0.5</td>
<td>40</td>
</tr>
</tbody>
</table>

Figure 4.6 RELAP5 SBWR600 Simulation for SP Separated-effect Test
4.1.4 SP Separated-effect Test Results[29]

As that has been mentioned before, SP separated-effect test boundary conditions are DW pressure, SP water level, RPV heater power (steam flow rate), and the air mass flow rate. The SP water level was measured by the differential pressure transducers. The air mass flow rate was controlled by air mass flow controller. The DW pressure and the RPV heater power during a test (test ID: LSF1, DW pressure 230 kPa, steam flow rate 70 g/s, air concentration 0%) are shown in Figure 4.7 and Figure 4.8.

In the following figures, thermal stratification is defined as the difference between the average of thermocouple readings at the highest level (Level 1 in Figure 4.3) and the average of thermocouple readings at the lowest level (Level 8 in Figure 4.3).

1). SP Initial Temperature Effect

Generally no noticeable effect of the pool initial temperature on the pool thermal stratification was observed, except for the high initial pool temperature case (60 °C), in which the thermal stratification is smaller than the other cases. The reason is that the surface water will be quickly heated up to the saturation temperature in the high initial pool temperature case (Figure 4.9).

2). Drywell Pressure Effect

There is small effect of the DW pressure on the pool thermal stratification. Larger temperature difference between the top of pool and the bottom of pool has been observed at high DW pressure. This may be explained by the air bubble size and the pool recirculation pattern will be affected by the DW pressure (Figure 4.10).
3). Steam Flow Rate Effect

The steam flow rate affects the pool thermal stratification. High steam flow rate makes more pool thermal stratification. This is reasonable because in high steam flow rate cases more energy was discharged into the pool (Figure 4.11).

4). Vent Opening Submergence Depth Effect

The vent opening affects the pool thermal stratification. The vent opening at the higher level will have more degree of pool thermal stratification than the vent opening at the lower level with the same other conditions. This is easy to understand because vent opening at the lower level means more pool liquid to participate in the energy absorption process (Figure 4.12).

5). Thermal Stratification for Pure Steam Injection Cases

Several tests were performed to investigate the thermal stratification, direct contact condensation, and the pool circulation driven by thermal plume and/or steam jet in the SP. The axial temperature distribution at $R = 106 \text{ cm}$ (the outermost ring of thermocouple location in Figure 4.3) is plotted in Figure 4.13. This figure shows that the pool region above the vent opening mixes homogeneously, while the pool region below the vent opening stays at the initial temperature. This means that heat is accumulated only in the upper pool region which is above the elevation of vent opening. In other words, thermal stratification reduces the effective pool inventory available for energy storage.

Two dimensional temperature distributions are plotted in Figure 4.14. This figure shows that high temperature steam condenses immediately at the vent exit. The heated-up liquid forms rising plume that is driven by the buoyancy force. Noticeable thermal stratification appears between the upper portion of the pool and the lower portion of the
pool around 500 seconds. As the time goes by, the energy starts to transfer from the upper pool to the lower pool through heat conduction and pool recirculation driven by rising plume. Figure 4.15 shows a sketch on pool circulation driven by the rising hot water plume.

6). Thermal Stratification for Steam/Air Mixture Injection Cases

The axial temperature distribution at $R = 106$ cm (the outermost ring of thermocouple location in Figure 4.3) is shown in Figure 4.16. This figure shows that complete pool mixing has been achieved in the high noncondensable gas injection rate cases. The pool is uniformly heated up by the condensed steam.

Two dimensional temperature distributions are plotted in Figure 4.17. This figure shows that high temperature steam condenses immediately at the vent exit. The heat-up liquid and noncondensable gas form rising bubble plume. The surrounding pool liquid is circulated not only driven by the hot liquid plume caused by the liquid temperature difference, but also driven by the rising gas bubble plume caused by the gas-liquid density difference. This pool recirculation flow pattern is shown in Figure 4.18. Due to the strong entrainment effect of the bubble plume, the entire pool is well mixed and thermal stratification in the pool disappeared.
Figure 4.7 RPV, DW and SP Pressure during the LSF1 Test

Figure 4.8 RPV Heater Power during the LSF1 Test
Figure 4.9 Effect of Initial Pool Temperature on SP Thermal Stratification

Figure 4.10 Effect of DW Pressure on SP Thermal Stratification
Figure 4.11 Effect of Steam Flow Rate on SP Thermal Stratification

Figure 4.12 Effect of Vent Opening Submergence Depth on SP Thermal Stratification
Figure 4.13 Pool Temperature at R = 106 cm for 0.07 kg/s Pure Steam Injection Case
Figure 4.14 2-D Temperature Distributions during Pure Steam Injection
Figure 4.15 Schematic of Pool Circulation Driven by Thermal Plume

Figure 4.16 Pool Temperature at R = 106 cm for 0.07 kg/s Steam and 3.684 g/s Air
Figure 4.17 2-D Temperature Distributions during Steam and High Noncondensable Gas Injection Case
Figure 4.18 Schematic of Pool Circulation Driven by Thermal and Air Bubble Plume
4.2 RELAP5 Simulation for the Suppression Pool Mixing and Condensation

The RELAP5 simulation for the suppression pool mixing and condensation can be performed based on the test data. The mixing process can be simulated in the code by applying parallel pipe components connected with cross flow junctions. Minor loss values at the flow junctions should be carefully chosen in order to let the code have the suitable mixing process. The RELAP5 nodalization for simulating the SP mixing and condensation is shown in Figure 4.19. Some preliminary results are shown in Figure 4.20 and Figure 4.21.

This approach is purely empirical because arbitrary K values have been chosen for flow junctions to affect the code predicted temperature values at different levels.

Figure 4.19 Nodalization of RELAP5 for PUMA Mixing and Condensation
Figure 4.20 RELAP5 Predicted Temperature Profile for Pure Steam Injection Case

(Steam Flow Rate = 0.1 kg/s)

Figure 4.21 RELAP5 Predicted Temperature Profile for Steam/Air Injection Case

(Steam Flow Rate = 0.1 kg/s, Air Concentration 5%)
CHAPTER 5. SUPPRESSION POOL MIXING ANALYSIS

5.1 Modeling Method

This chapter discusses the theoretical analysis of the suppression pool mixing process caused by a single phase turbulent buoyant jet/plume, or by a two-phase turbulent buoyant jet/plume. Section 5.2 defines the problem and discusses the general modeling approach for an underwater turbulent buoyant jet/plume. Section 5.3 discusses the treatment of entrainment rate in order to model the mixing process for free shear flows. Then in Section 5.4 the integral modeling method is applied to analyze the single phase axisymmetric turbulent buoyant jet/plume which is generated by the condensed steam inside the suppression pool. The application of this integral modeling method on the two-phase jet/buoyant plume, which is generated by the injection of noncondensable gas with steam into the suppression pool, has been discussed in Section 5.5. The results predicted by the model are compared with the available experimental data.

5.2 Governing Equations for Turbulent Buoyant Jets and Plumes

Turbulent jets and plumes are turbulent flows produced by the momentum and buoyancy sources. The jet flow is dominated by a continuous source of momentum, while the plume flow is dominated by a continuous source of buoyancy. An example of the jet flow is the hot gas discharging from an airplane engine. In this case the momentum inertia
is so huge that the momentum generation by buoyancy is not important in the analysis of the flow. On the other hand, in the analysis of the smoke generated from a burning cigarette the buoyancy effect should not be ignored.

Both the momentum inertia and buoyancy effect should be considered in the analysis of the fluid flow produced by the injection of steam or steam/noncondensable gas into suppression pool. The momentum inertia controls flow pattern in the region close to the injection point, however, the buoyancy effect becomes important along the flow path and eventually will dominate the far region from the injection point. The fluid motion in such case will be governed by the inertial, buoyant and viscous forces. The non-dimensional numbers that can be utilized to characterize the flow conditions are summarized here.

The Reynolds number describes the relative ratio between the inertial and the viscous force. At the injection point, the jet Reynolds number can be defined as in Eq. 5.1, where \( U_0 \) is the mean jet flow velocity at the injection point, and \( D_0 \) is the jet equivalent flow diameter at the injection point which usually equals to the diameter of the opening,

\[
\text{Re} = \frac{U_0 D_0}{\nu} \tag{Eq. 5.1}
\]

The Grashof number describes the relative ratio between the buoyant force and the viscous force. In Eq. 5.2, the definition of Grashof number at the jet injection point is given. Here, \( \rho_0 \) is the water density at the jet injection point, and \( \rho_a \) is the pool water density at the same level,
The densimetric Froude number given in Eq. 1.1 defines the ratio of inertial to buoyant force. It is an important parameter which can be utilized to define different regions along the jet/plume flow path in the suppression pool. Figure 5.1 shows different regions along a turbulent jet/plume flow. The Froude number is large in the non buoyant region, which is close to the jet injection point. The Froude number decreases along the jet/plume flow path, and becomes small in the buoyant region which is far away from the injection point. Both the momentum inertia and buoyant force are important in the intermediate region.

There are three approaches that usually have been utilized to predict the turbulent buoyant jet/plume flow. The most straightforward way is to correlate experimental data with the help of dimensionless study. This method can generate useful guild lines for the future engineering application, but the validity of such correlations is limited to such cases that should have similar boundary conditions and initial conditions as those in the experiment.

Another approach is to utilize the integral modeling method. In this approach first conservation equations of mass, momentum and energy should be setup for the jet/plume flow. Then the empirical profile shapes, usually a Gaussian profile or a “top-hat” profile, for velocity, temperature or concentration across the jet should be made so that the integration of the conservation equations over the cross section of the jet/plume flow path is possible. Such integration can transfer the conservation equations from partial differential equations (PDEs) to ordinary differential equations (ODEs) which describe the
mass, momentum and energy variation along the jet/plume flow path. An additional equation which describes the entrainment rate of the ambient fluid into the jet/plume should be given in order to close the equation set of the problem. The development of such entrainment model will be discussed in Section 5.4. With the proper entrainment model, the integral modeling method can generate reasonable prediction of the major jet/plume flow characterization parameters, such as the centerline velocity, temperature or concentration along the flow path.

Figure 5.1 Turbulent Buoyant Jet/Plume Flow[30]
The last approach is to directly solve the same PDEs of the conservation equations as in the integral modeling method, with the proper assumptions regarding the turbulent processes which can describe the local turbulent fluxes of momentum, energy or species. This approach does not need to assume empirical profile shapes of the jet/plume flow and the entrainment rate along the jet/plume path, but can calculate these profiles and the entrainment rate as a part of the solution. However, due to the complexity of the turbulence modeling, the final set of PDEs usually can only be solved by using numerical method.

The general governing equations for a steady state axisymmetric two-dimensional round turbulent jet can be described in the following equation set. The flow is modeled in the cylindrical axes \((r,z)\) with \(z\) axis extends along the axis of flow, and velocity components is described as \((u,v)\) with \(u\) component along the \(z\) axis. The gravity vector opposes the \(z\) coordinate. The axisymmetric jet flow is assumed to be injected from a discharging nozzle of diameter \(D_0\), with initial velocity \(W_0\), and temperature \(T_0\). The initial jet fluid density \(\rho_0\) is associated with the initial temperature \(T_0\).

The continuity equation:

\[
\frac{1}{r} \frac{\partial (\rho vr)}{\partial r} + \frac{\partial (\rho u)}{\partial z} = 0 \tag{Eq. 5.3}
\]

The \(z\)-momentum equation:

\[
\frac{1}{r} \frac{\partial (\rho uv)}{\partial r} + \frac{\partial (\rho u^2)}{\partial z} = -g (\rho - \rho_\infty) - \frac{1}{r} \frac{\partial (\rho u \overline{v'})}{\partial r} \tag{Eq. 5.4}
\]
The thermal energy equation:

\[
\frac{1}{r} \frac{\partial (\rho r v T)}{\partial r} + \frac{\partial (\rho u T)}{\partial z} = - \frac{1}{r} \frac{\partial (\rho r v T')}{\partial r}
\]  
Eq. 5.5

Here, if the thermal expansion coefficient \( \beta_T \) is defined as

\[
\beta_\infty = - \left. \frac{1}{\rho} \frac{\partial \rho}{\partial T} \right|_{r_\infty}
\]  
Eq. 5.6

With the Boussinesq approximation is applied to the z-momentum equation, the gravity term in Eq. 5.4 can be expressed by

\[
-g (\rho - \rho_\infty) = \rho g \beta_\infty (T - T_\infty)
\]  
Eq. 5.7

Thus, the z-momentum equation can be written as

\[
\frac{1}{r} \frac{\partial (\rho r u v)}{\partial r} + \frac{\partial (\rho u^2)}{\partial z} = \rho g \beta_\infty (T - T_\infty) - \frac{1}{r} \frac{\partial (\rho r u v')}{\partial r}
\]  
Eq. 5.8
The boundary conditions for jet flow with a constant injection rate can be listed as

\begin{align*}
  u(r,0) &= u_0 & u'(r,0) &= v'(r,0) = T'(r,0) = 0 \\
  u(r,\infty) &= 0 & u(r,\infty) &= 0 \\
  v(0,z) &= 0 & u(\infty,z) &= u'(\infty,z) = 0 \\
  T(r,0) - T_\infty &= \Delta T_0 \\
  T(r,\infty) - T_\infty &= 0
\end{align*}

Eq. 5.9

Above equations show that in order to predict the characterization parameters in the buoyant turbulent jet flow, proper models for turbulent fluxes should be given along with the boundary conditions and initial conditions.

5.3 Integral Modeling of the Turbulent Buoyant Jets

The conservative equations that describe the turbulent buoyant jets can be integrated over the cross-section to yield the integral governing equations for the jet flow. The integration of the continuity equation (Eq. 5.3) with the boundary conditions (Eq. 5.9) yields the liquid entrainment rate from the ambient to the jet region

\[
  \frac{d}{dz} \int_0^{R_j} 2\pi r u \rho dr = -(2\pi \rho v)_{r=\infty} = E
\]

Eq. 5.10

Here, \( R_j \) is the jet radius, and \( E \) is the entrained mass per unit length of the jet. By this definition, we have \( u(r > R_j, z) = 0 \) and \( T(r > R_j, z) = T_\infty \).
The integration of the z-momentum equation (Eq. 5.4) with the boundary conditions (Eq. 5.9) yields the integral z-momentum equation

\[
\frac{d}{dz} \int_0^R \rho ru^2 dr = g \int_0^R r (\rho_\infty - \rho) dr
\]  
Eq. 5.11

The integration of the thermal energy equation (Eq. 5.5) with the boundary conditions (Eq. 5.9), combining the integral continuity equation (Eq. 5.10), yields the integral thermal energy equation

\[
\frac{d}{dz} \int_0^R \rho ru (T - T_\infty) dr = - \frac{dT_\infty}{dz} \int_0^R \rho u dr
\]  
Eq. 5.12

The growth of the jet/plume is achieved by an inflow of the ambient fluid, which is described in Eq. 5.10. The entrainment hypothesis first suggested by Morton et al.\textsuperscript{[31]} states that the mean inflow velocity across the edge of the turbulent flow is assumed to be proportional to the characterization jet velocity, usually the local centerline velocity or the cross-section averaged mean velocity. The entrainment velocity \( v_E \) given by Monton et al. thus can be written as

\[
v_E = \left| v_{R_t} \right| = \alpha u_m
\]  
Eq. 5.13
Here, $\alpha$ is the entrainment coefficient, and $u_m$ is the jet flow centerline velocity. Values for $\alpha$ has been given in the literature. Fischer et al.\cite{32} have utilized the experimental data to show that for round jet/plume flow

$$\alpha_{jet} = 0.054$$
$$\alpha_{plume} = 0.083$$

Eq. 5.14

Here the jet flow has been assumed to have the Gaussian velocity profile

$$u = u_m \exp \left[-\left(\frac{r}{R_z}\right)^2\right]$$

Eq. 5.15

These values are very close to the entrainment coefficient values suggested by Cederwall\cite{33}, Chan et al.\cite{34} and Henderson\cite{35}, where the entrainment coefficients for round jet/plume flow are given as

$$\alpha_{jet} = 0.057$$
$$\alpha_{plume} = 0.082$$

Eq. 5.16

The entrainment coefficient values for a top-hat velocity profile should be modified from the above suggested value by multiplying $\sqrt{2}$. This kind of treatment on entrainment is simple and has been proved successful for the flows have similar turbulent structure and balance of forces along the jet/plume height. However, it is now generally
agreed that the entrainment coefficient should be modeled as a function of the local level of turbulence. List et al.\textsuperscript{[36]} suggests that for homogenous fluids the entrainment coefficient should be correlated to the local densimetric Froude number

\[
\alpha = \frac{C_1}{2} \left( 1 + \frac{2}{3} \frac{F_{rp}^2}{F_r^2} \right) \quad \text{Eq. 5.17}
\]

Here, \( F_r \) is the densimetric Froude number which is defined in Eq. 1.1. The values of \( C_1 \) and \( F_{rp}^2 \) are given by Henderson\textsuperscript{[35]} as

\[
C_1 = 0.102 \quad \text{Eq. 5.18}
\]
\[
F_{rp}^2 = 16.5
\]

An alternative representation for the definition of \( \alpha \) in Eq. 5.17 can be given as

\[
\alpha = \alpha_{jet} + \left( \alpha_{plume} - \alpha_{jet} \right) \frac{F_{rp}^2}{F_r^2} \quad \text{Eq. 5.19}
\]

Wu et al.\textsuperscript{[37]} compared experimental data and concluded that

\[
E = 2\pi R c_x u_m \left( 0.057 + \frac{0.4775}{F_r^2} \right) \quad \text{Eq. 5.20}
\]
Lee et al.\textsuperscript{[38]} utilized a similar shear entrainment coefficient in their Lagrangian buoyant jet model

\[ \alpha = \sqrt{2} \left( 0.057 + \frac{0.554}{F_r^2} \right) \quad \text{Eq. 5.21} \]

Here, \( \sqrt{2} \) is multiplied because Lee’s model utilizes top-hat velocity profile in the jet flow.

### 5.4 Modeling Mixing inside Suppression Pool with Pure Steam Injection

The PUMA suppression pool separated-effect test results presented in Figure 4.13 and Figure 4.14 show that for the pure steam injection case, the pool mixture is driven by the buoyant thermal jet/plume which is generated by the condensed steam. The photo images of the injection port shown in Figure 5.2, which are taken during the test by the high speed camera also prove that the injected steam will be completely condensed very quickly after it exists the injection port. Figure 4.15 illustrates the imaginary pool mixing process based on the temperature measurement results.

In view of these facts, the fully condensation of the injected steam at the exit of the injection port has been assumed in the modeling. Thus this model intends to calculate the pool mixing driven by the buoyant thermal jet/plume. The calculation domain that is considered in the model is shown in Figure 5.3. The injection port elevation is 0.98 m from the bottom of tank, and the initial water level in the suppression pool is 1.55 m. The tank equivalent diameter is 2.794 m.
Figure 5.2 Test Image of Injection Port (Pure Steam Injection Case)

Figure 5.3 Calculation Domain for the Pure Steam Injection Case
The approach that is similar to Peterson et al.\cite{15}, Peterson\cite{39} and Christensen et al.\cite{40} has been adopted to model the mixing process inside the suppression pool. The pool is divided into two zones, say, the fluid contained within the buoyant jet/plume and the fluid in the ambient volume. The fluids in the two zones are coupled with each other through the entrainment of the ambient fluid into the buoyant jet/plume as shown in Figure 5.4.

![Figure 5.4 One-dimensional Integral Model for Buoyant Jet/Plume](image)

From the jet/plume injection starting time to the jet/plume reaches the surface of the pool, the thermal jet/plume zone will entrain the ambient fluid, but the fluid in the pool zone will not feel the thermal effect of the jet/plume due to the fact that the temperature of the fluid flows from the upper level of the pool still is same as the pool initial temperature. The heat up of the pool fluid will start after the hot fluid inside the jet/plume reaches the pool surface. A Lagrangian buoyant jet model that is similar to J.H.W. Lee et al.\cite{38} is utilized to predict the jet/plume parameters in this flow establishing period.
The jet/plume is treated as consisting of a sequential series of jet/plume elements. Each jet/plume element is characterized by its location, average velocity, temperature, width and thickness. The model discusses the jet/plume element locates at \( z_k \) with velocity \( u_k \) at the \( k \)th time step. The temperature, density and jet/plume radius are denoted by \( T_k \), \( \rho_k \) and \( r_k \). The element thickness is denoted by \( h_k \) and by definition is proportional to the magnitude of the local jet velocity, i.e., \( h_k \propto u_k \). The turbulent entrainment of the ambient fluid into the plume element is calculated at each step. The mass, momentum, energy conservation equations can be solved based on the proper entrainment model.

The model can be written in the following equation sets:

**Mass conservation equation**

\[
M_{k+1} = M_k + \Delta M_k \\
M_{k+1} = \rho_{k+1} \pi r_{k+1}^2 h_{k+1}
\]

Eq. 5.22

**Temperature and density equation**

\[
T_{k+1} = \frac{M_k T_k + \Delta M_k T_{k+1}}{M_{k+1}}
\]

\[
\rho_{k+1} = \rho(T_{k+1}, P_{k+1})
\]

Eq. 5.23

**Z-momentum equation**
\[ u_{k+1} = \frac{M_k u_k + M_{k+1} \left( \frac{\Delta \rho}{\rho} \right)_{k+1} g \Delta t}{M_{k+1}} \]  

Eq. 5.24

Thickness and radius of jet/plume elements

\[ h_{k+1} = \frac{V_{k+1} h_k}{V_k} \]  

Eq. 5.25

\[ r_{k+1} = \left( \frac{M_{k+1}}{\rho_{k+1} \pi h_{k+1}} \right)^{1/2} \]

Location of the jet/plume elements

\[ z_{k+1} = z_k + u_{k+1} \Delta t \]  

Eq. 5.26

The initial conditions

\[ u_0 = U_o \]  

\[ r_0 = 0.5D_o \]  

\[ T_0 = T_o \]  

Eq. 5.27

The setting of the initial jet/plume velocity in the current model is based on the assumption that all injected steam will be condensed immediately when it contacts with the suppression pool cold water. Only saturated water will flow out of the injection port.
The hot water injection rate at the exist can be calculated from the energy conservation equation

\[ \dot{m}_s h_{g,sat} = \dot{m}_f h_{f,sat} + \dot{m}_j \left( h_{f,sat} - h_{f,ini} \right) \]  

Eq. 5.28

Here, \( \dot{m}_s \) is the mass flow rate of the steam, which is the known boundary condition in the suppression pool tests. The injection velocity of the saturated water can be calculated from the mass flow rate and the injection port diameter

\[ u_0 = \frac{\dot{m}_s + \dot{m}_f}{\rho_{f,sat} A_0} \]  

Eq. 5.29

Thus all necessary initial conditions for the jet/plume flow can be specified once the test boundary conditions are known. The entrainment rate should be given in order to calculate close the problem.

\[ \Delta M_k = k \alpha 2 \pi r_k h_k u_k \Delta t \]  

Eq. 5.30

After the thermal jet/plume reaches the surface of the suppression pool, the energy equation for the ambient water should also be considered. From this time point, the one-dimensional Eulerian conservation equations for both the jet/plume fluid and the ambient fluid are utilized in the model. The equation sets can be written as the followings:
Mass conservation equation for jet/plume

\[ M_{k+1,j+1} = M_{k,j} + \Delta M_{k,j} + \left( \rho_j u_j A \right)_{k-1,j} \Delta t - \left( \rho_j u_j A \right)_{k,j} \Delta t \]  
\[ M_{k+1} = \rho_{k+1} \pi r_{k+1}^2 h_{k+1} \]  

\[ \text{Eq. 5.31} \]

Energy conservation equation for jet/plume

\[ T_{k,j+1} = \frac{\left[ \left( dt \cdot (\rho_j u AT)_{k-1,j} \right) \left( dt \cdot (\rho_j u AT)_{k,j} \right) \right]}{M_{k,j+1}} + \left( \Delta ET_p \right)_{k,j} + \left( MT \right)_{k,j} \]  
\[ \text{Eq. 5.32} \]

Energy conservation equation for ambient fluid

\[ T_p \bigg|_{k,j+1} = \left( M_p \bigg|_{k,j} - \sum_{i=0}^{\Delta t} \Delta E_{i,j} \right) T_p \bigg|_{k,j} + \sum_{i=0}^{\Delta t} \Delta E_{i,j} T_p \bigg|_{k+1,j} + E_{\text{cond}} \]  
\[ \text{Eq. 5.33} \]

\[ E_{\text{cond}} = k \left[ \left( T_p \bigg|_{k+1,j} - T_p \bigg|_{k,j} \right) A_p \bigg|_{k+1,j} - \left( T_p \bigg|_{k,j} - T_p \bigg|_{k-1,j} \right) A_p \bigg|_{k,j} \right] \Delta t \]  
\[ \text{Eq. 5.33} \]
Momentum conservation equation for jet/plume

\[
    u_{k,t+1} = \frac{B - M_{\text{cov}} + (\rho_f u)_{k,t}}{\rho_f_{k,t+1}}
\]

\[
    M_{\text{cov}} = \frac{\Delta t}{dz} \left[ \left( \rho_f uuA \right)_{k,t} - \left( \rho_f uuA \right)_{k-1,t} \right]
\]

\[
    B = \left( \rho_p - \rho_f \right)_{k,t} u_{k,t} g \Delta t
\]

Eq. 5.34

The entrainment coefficient can be chosen from the values listed in Section 5.3.

Figure 5.5 Prediction Results vs. Test Data for Steam Injection as 0.07 kg/s
The comparison between the numerical model predictions with the experiment data on 0.07 kg/s and 0.12 kg/s steam injection flow rate cases have been shown in Figure 5.5 and Figure 5.6. In where the solid lines represent the numerical predictions, and the dot lines represent the experiment data. It can be seen from the comparison that the model predictions have relative large error on the initial stages, but the prediction gets closer to the data in the later phase. This may be caused by the fact that the entrainment model utilized here may underestimate the global circulation caused by the thermal jet/plume, thus the injected energy will first accumulate near the surface of pool, then slowly transfer down to the lower part of the pool.
5.5 Modeling Mixing inside Suppression Pool with Air Injection

The results of suppression pool mixing and condensation separated-effect test clearly show that pool mixing phenomena is dominated by the noncondensable gas. Figure 4.16 and Figure 4.17 show that for the steam/air mixture injection case, the pool mixture is enhanced by the two-phase buoyant jet. The photo images of the injection port shown in Figure 5.7, which are taken during the test by the high speed camera, also show that a two-phase buoyant jet exists after the injection port. Figure 4.18 illustrates the imaginary pool mixing process based on the temperature measurement results.

Figure 5.7 Test Image of Injection Port (Steam/Air Injection Case)
The one dimensional integral model presented in the previous section should be modified to account for the two phase flow effect. The jet density should be replaced by the mean density of the two phase flow.

\[
\rho_m = \alpha \rho_g + (1-\alpha) \rho_f \tag{Eq. 5.35}
\]

Here, \( \alpha \) is the local void fraction. The value of \( \alpha \) can be estimated by using the drift flux model.

Except for the jet/plume density modification, a global recirculation velocity in the pool zone should also be specified in order to account for the strong pumping effect of the air bubble on the pool liquid. The following integral momentum equation for two phase flow can be utilized to estimate this global recirculation rate.

\[
\frac{\rho_m u^2 f_{jet}}{2} + \frac{\rho_f u^2}{2} \frac{f(H-l_{jet})}{d} + \frac{\rho_f u^2}{2} \frac{fH}{d} = (\rho_f - \rho_m) g l_{jet} \tag{Eq. 5.36}
\]

Here, it is assumed that the driving head for the recirculation flow comes from the buoyant force caused from the density difference of the pool water and two phase jet. It is also assumed that the pool recirculation flow occurs in the near wall region that has the same diameter of the jet flow. Validity of these assumptions should be checked by the experimental data or by the CFD numerical simulations.
CHAPTER 6. NUMERICAL MODELING OF THE PUMA SP TEST

The discussion on the theoretical modeling of suppression pool mixing shows that the simple one-dimensional integral modeling method can reasonably predicts results of the pure steam injection case, however, it has some difficult to predict the results of the steam/air injection cases. A good model which can describe the air bubble pumping effect on the pool liquid which locates below the injection port level should be incorporated in order to explain the whole pool mixing observed in the experiment. The experiment data for pool fluid local velocity at different \((r, z)\) locations should be provided in order to build such model. However, due to the limitation of the available instrumentation inside the PUMA suppression pool, such experiment data is unavailable. Therefore, a three dimensional numerical model of the PUMA suppression pool under the steam/air injection conditions have been developed by using FLUENT 6.2, which is a commercially available Computational Fluid Dynamics (CFD) package that is capable to perform the multi-dimensional, multi-phase numerical calculation in a pre-set calculation domain.
Transport Equations and Turbulence Models in FLUENT

FLUENT 6.2 utilizes the Navier-Stokes equations to describe the processes of mass, momentum and heat transfer. In the Eulerian multiphase model, FLUENT 6.2 solves the following equation sets.

The mass conservation equation for phase \( q \) can be written as,

\[
\frac{\partial (\alpha_q \rho_q \bar{u}_q)}{\partial t} + \nabla (\alpha_q \rho_q \bar{u}_q) = \sum_{p=1}^{n} (\dot{m}_{pq} - \dot{m}_{qp}) + S_q
\]

Here \( \dot{m}_{pq} \) models the mass transfer rate from phase \( p \) to phase \( q \), \( S_q \) is the volumetric mass source of phase \( q \), and \( \alpha_q \) is volumetric fraction of phase \( q \).

The momentum conservation equation for phase \( q \) can be written as,

\[
\frac{\partial (\alpha_q \rho_q \bar{u}_q)}{\partial t} + \nabla (\alpha_q \rho_q \bar{u}_q) = -\alpha_q \nabla p + \nabla \bar{e}_q + \alpha_q \rho_q \bar{g} \\
+ \sum_{p=1}^{n} (\bar{R}_{pq} + \dot{m}_{pq} \bar{u}_p - \dot{m}_{qp} \bar{u}_q) + (\bar{F}_q + \bar{F}_{lift,q} + \bar{F}_{vsm,q})
\]

Here \( \bar{R}_{pq} \) denotes the interfacial force between phases, and the \( q^{th} \) phase stress tensor \( \bar{\tau}_q \) is related to the strain rate by
\[ \bar{\tau}_q = \alpha_q \mu_q \left( \nabla \bar{u}_q + \nabla \bar{u}_q^T \right) + \alpha_q \left( \lambda_q - \frac{2}{3} \mu_q \right) \nabla \cdot \bar{u}_q \bar{I} \]  \hspace{1cm} \text{Eq. 6.3}

The energy conservation equation for phase \( q \) can be written as,

\[ \frac{\partial (\alpha_q \rho_q h_q)}{\partial t} + \nabla (\alpha_q \rho_q \bar{u}_q h_q) = -\alpha_q \frac{\partial p_q}{\partial t} - \nabla \bar{q}_q + S_q + \bar{\tau}_q : \nabla \bar{u}_q \]

\[ + \sum_{p=1}^{n} \left( \bar{Q}_{pq} + \dot{m}_{pq} h_{pq} - \dot{m}_{qp} h_{qp} \right) \]  \hspace{1cm} \text{Eq. 6.4}

Here \( S_q \) denotes the energy source with the unit of \([ML^{-1}T^{-3}]\).

FLUENT 6.2 provides five options for modeling the turbulence of multiphase flows. The \( k - \varepsilon \) mixture turbulence model has been utilized in the present work, because the \( k - \varepsilon \) two-equation model has been proven to be stable and numerically robust and has a well-established regime of predictive capability. The \( k - \varepsilon \) two-equation model utilizes the gradient diffusion hypothesis to relate the Reynolds stresses to the mean velocity gradients and the turbulent viscosity. The turbulent viscosity is modeled as the product of a turbulent velocity and turbulent length scale. In two-equation models, the turbulence velocity scale is computed from the turbulent kinetic energy, which is provided from the solution of its transport equation. The turbulent length scale is estimated from two properties of the turbulence field, usually the turbulent kinetic energy and its dissipation rate. The dissipation rate of the turbulent kinetic energy is provided from the solution of its transport equation. In the \( k - \varepsilon \) model, \( k \) denotes the turbulent kinetic energy and is
defined as the variance of the fluctuations in velocity, and $\varepsilon$ denotes the turbulence eddy dissipation.

The transport equations for the $k-\varepsilon$ mixture multiphase model in FLUENT 6.2 are introduced as following:\cite{42}.

The transport equation for mixture turbulent kinetic energy can be written as,

$$\frac{\partial}{\partial t}(\rho_m k) + \nabla(\rho_m \bar{u}_m k) = \nabla \left( \frac{\mu_{t,m}}{\sigma_k} \nabla k \right) + G_{k,m} - \rho_m \varepsilon$$

Eq. 6.5

The mixture turbulence eddy dissipation equation:

$$\frac{\partial}{\partial t}(\rho_m \varepsilon) + \nabla(\rho_m \bar{u}_m \varepsilon) - \nabla \left( \frac{\mu_{t,m}}{\sigma_s} \nabla \varepsilon \right) = \frac{\varepsilon}{k} \left( C_{1\varepsilon} G_{k,m} - C_{2\varepsilon} \rho_m \varepsilon \right)$$

Eq. 6.6

Here $\rho_m$ is the mixture density, which can be calculated from

$$\rho_m = \sum_{q=1}^{n} \alpha_q \rho_q$$

Eq. 6.7

And $\bar{u}_m$ is the mixture velocity, which can be calculated as

$$\bar{u}_m = \frac{\sum_{q=1}^{n} \alpha_q \rho_q \bar{u}_q}{\rho_m}$$

Eq. 6.8
It is assumed in the $k-\varepsilon$ model that the turbulence viscosity can be linked to the turbulence kinetic energy and dissipation through the relation

$$\mu_{t,m} = C_\mu \rho_m \frac{k^2}{\varepsilon}$$

Eq. 6.9

$G_{k,m}$ is the production of turbulence kinetic energy, which is modeled by

$$G_{k,m} = \mu_{t,m} (\nabla \bar{u}_m + \nabla \bar{u}_m^T) : \nabla \bar{u}_m$$

Eq. 6.10

The default empirical model constants are set as

$$C_\mu = 0.09, \ C_{\varepsilon_1} = 1.44, \ C_{\varepsilon_2} = 1.92, \ \sigma_k = 1.0, \ \sigma_\varepsilon = 1.3$$

$$\sigma_\rho = 0.9, \ C_3 = 1.0$$

Eq. 6.11

6.2 PUMA Suppression Pool Mixing Calculation in FLUENT

Figure 6.1 show the calculation domain in FLUENT for simulating the PUMA suppression pool separated-effect tests. Only one third of the pool is modeled since the test facility geometry and boundary conditions are periodically repeating over three 120 degree zones, with each zone contains one vertical vent opening. The calculation domain contains the water pool and the gas space. The inner cylinder, outer cylinder, bottom face and top face are modeled as wall boundaries. A vertical vent opening at the inner cylinder surface and an outflow opening at the top wall have also been included in the model. Steam and
air mixture enters the calculation domain through the vertical vent opening, and air leaves the calculation domain through the top wall opening.

Hexahedral meshing is applied to this suppression pool calculation model. Fine meshes have been given near the inner cylinder wall, the outer cylinder wall, and the pool surface. Relative large meshes have been given for zones far away from the cylinder walls. Average mesh size is about 50 mm. Total cell number inside this calculation mesh is close to 120,000.

The CFD calculation has been performed for 2 test cases, i.e., test LSF1 and test LSF4 in Table 4.4. CFD calculation conditions are listed in Table 6.1. The Eulerian multiphase model has been enabled for suppression pool mixing calculation. Three phases have been enabled in the CFD calculation model. Water is assigned as the primary phase, vapor is assigned as the secondary phase, and air is assigned as the third phase. Constant fluid properties, which are evaluated at the averaged pool temperature and pressure, have been assigned in the calculation model. However, water density has been set up as a function of temperature, in order to simulate the buoyancy effect of heated-up water. Based on the flow visualization result (Figure 5.2), air and steam flow in the water pool has been assumed to be in the bubbly flow regime. The $k – \varepsilon$ mixture turbulence model for mixture phase, combined with the standard wall functions, has been enabled in the suppression pool mixing calculation.

Vapor condensation rate in this CFD calculation has been modeled as the function of void fraction, temperature difference between vapor phase and water phase, and the available interfacial area in current cell,
\[ \dot{m}_{\text{cond}} = \frac{6 \times \alpha_{\text{fg}}}{d_0} h_{\text{conv}} \frac{T_{\text{sat}} - T_f}{l_{\text{fg}}} \]  

Eq. 6.12

Here, \( h_{\text{conv}} \) is the convection heat transfer rate from vapor to interface, and \( d_0 \) is typical vapor bubble size. Above equation has been implemented into FLUENT calculation model by utilizing user-defined functions.

<table>
<thead>
<tr>
<th>Test No.</th>
<th>Drywell Pressure (kPa)</th>
<th>Steam Injection Rate (g/s)</th>
<th>Air Concentration (%)</th>
<th>SP Initial Temperature (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>LSF1</td>
<td>230</td>
<td>70</td>
<td>0</td>
<td>50</td>
</tr>
<tr>
<td>LSF4</td>
<td>230</td>
<td>70</td>
<td>5</td>
<td>50</td>
</tr>
</tbody>
</table>

Two cases start from same initial conditions, such as, initial water pool height as 1.72 m, and initial water and gas space temperature as 50 °C (Figure 6.2). At time 0 sec, steam or steam/air mixture starts to be injected through the vertical vent opening.

Temperature calculation results in LSF1 (pure steam injection) are shown in Figure 6.3. At initial stage, pool temperature is set as 50 °C. Pool temperature increases after steam injection starts. At 300 seconds, thermal stratification has been developed and pool surface temperature has reached 56 °C, while water temperature in zone lower than injection port slowly increases due to conduction and thermal diffusion.
Figure 6.1 SP Modeling in FLUENT

Figure 6.2 Initial Status for LSF1 and LSF4 Calculation
At 2700 seconds, CFD calculation predicts pool surface temperature will almost reach 70 °C. Water pool above the injection port almost has the same temperature. More thermal energy propagates to water that is lower than the injection port. These phenomena are same as what has been observed in the experiment.
The pool mixing pattern for LSF1 (pure steam injection) calculation, and for LSF4 (steam and air injection) calculation are shown in Figure 6.4 and Figure 6.5. Figure 6.4 shows that under the pure steam injection condition, an upwards plume forms after the
vertical vent injection port. The plume’s influence is limited to zones that are close to vertical vent pipe. After the plume reaches to the water surface, it spreads out along the water surface. Pool circulation and mixing under this case is not significant. On the other hand, Figure 6.5 shows strong pool circulation and mixing that is caused by the steam and air injection in LSF4 calculation. The pumping effect of air bubble plume has been successfully simulated in LSF4 calculation.

Since the SP separated-effect test mainly focused on evaluating the pool thermal stratification phenomenon, there is no attempting to measure the pool circulation velocity. However, pool circulation velocity is an important parameter for validating the CFD calculation results. Such data can help researchers to fine tuning the interfacial drag model, and the turbulence model in their calculation. In future study, some efforts should be paid to obtain pool circulation characteristic velocity.
CHAPTER 7. CONCLUSION

7.1 Summary of the Current Study

The main objective of this thesis is to assess the safety systems design of the next generation boiling water reactor through the analysis of the experiment data produced by the integral test facility and through the calculation of the best estimation reactor system safety code, such as RELAP5. To satisfy the above objectives, following areas have been studied and addressed: 1). discuss the PUMA main steam line break test data; 2). discuss the RELAP5 simulation results for both the SBWR and the PUMA main steam line break test; 3). discuss the important local phenomenon of suppression pool mixing and condensation; 4). model the suppression pool mixing process.

The safety systems design of the next generation boiling water reactor has been successfully demonstrated in the main steam line break test performed in a well-scaling down test facility, the PUMA facility. Test data demonstrates that under the postulated large break loss-of-accident condition, the automatic depressurization system can quickly release the RPV pressure thus short-term inventory injection from the gravity driven cooling system is possible to ensure core is always covered with cooling water. Long term core coverage is achieved by water injection from suppression pool. Core decay heat is removed from reactor mainly with the help of passive containment cooling system. Containment pressure in long-term cooling is kept below 260 kPa.
The scaling approach used for designing the PUMA facility has been checked and validated through RELAP5 code calculation. First, a RELAP5 code model has been built for simulating the main steam line break test of PUMA facility. Such facility level code is then validated directly with test data. RELAP5 calculation reasonably predicts the key data collected in the test, such as, the RPV pressure, the RPV water level, the GDCS injection rate, and the containment pressure. Second, a RELAP5 code model is built for prototype plant to predict the prototype plant behavior under the same postulated accident. Third, all collected test data are scaled-up to prototype plant level, by applying the same scaling laws for designing the test facility. Scaled-up test data then are compared to RELAP5 prototype plant calculation results. Good agreement has been found among them, which demonstrates the scaling approach used for designing the PUMA facility is successful.

Some uncertainties remain on RELAP5 predicting for the prototype plant behavior under the postulated accident conditions. One of these uncertainties is the thermal stratification in suppression pool after hot steam and air mixture blowing into it. This is due to RELAP5 essentially is a one-dimensional code, which is not suitable for predicting the pool circulation characteristics. Separated effect tests have been performed to investigate the pool mixing behavior under different steam and air discharging rate. Through the test, it is found that non-condensable gas discharging rate can significantly affect the thermal stratification in a large water pool. Detailed temperature measurement data have been obtained through these separated effect tests.

With the help of empirical correlations on jet/plume entrainment rate, the integral modeling method has been applied to predicting the suppression pool mixing process caused by a single phase turbulent buoyant jet/plume. Such simple model can reasonably
predict the measured pool temperature profile under the pure steam injection cases. However, this model has difficulty to predict the pool mixing enhanced by non-condensable gas injection.

A suppression pool CFD model has been built in FLUENT 6.2. The CFD model simulates one third of the test facility. Eulerian multiphase simulation has been performed, with steam condensation rate being calculated from user-defined functions. CFD calculation has been performed for pure steam injection case and for steam/air injection case. CFD calculation successfully reproduces the thermal stratification development under the pure steam injection condition. Predicted pool velocity field demonstrate that a weak plume is formed under the pure steam injection case, such plume only influence pool water in close distance. However, strong global recirculation is achieved in the CFD calculation for steam/air injection case. Air pumping effect has been predicted by the CFD calculation.

However, current CFD calculation model cannot be finely tuned due to the lack of velocity measurement data. Future works should be done for clarify following issues:

1). Collecting pool circulation characteristic velocity under different steam/air injection rate;

2). Benchmark the CFD calculation results with collected velocity data;

3). Fine tuning the interfacial drag force model, and the mixture turbulence model in the CFD calculation in order to accurately predict the pool mixing phenomena.
LIST OF REFERENCES
LIST OF REFERENCES


Figure A.1 RELAP5 Nodalization for SBWR RPV
Figure A.2 RELAP5 Nodalization for SBWR Containment and Vertical Vent
Figure A.3 RELAP5 Nodalization for SBWR PCCS
Figure A.4 RELAP5 Nodalization for SBWR ICS
Figure A.5 RELAP5 Nodalization for SBWR GDCS
Figure A.6 RELAP5 Nodalization for SBWR ADS and MSL
Figure A.7 RELAP5 Nodalization for SBWR GDCS Equalization Lines
Appendix B  Modeling for Suppression Pool Mixing in Matlab

% Model for pure steam case SP test
% Use Integral Method / Lagrangian Model
clear all;
close all;
% Define constant
PI = 3.1415926;
g = 9.8;
% Define SP geometry
D_pool = 2.817;
D_VV = 0.356;
eqD = (D_pool ^ 2 - D_VV ^ 2) ^ 0.5;
dt = 0.1; % dt can be chosen to be 0.1, 0.2, 0.5, 1.0 seconds for sensitivity study
z0 = 0.98; % Injection port height from the bottom of SP tank
zs = 1.54; % Water surface height from the bottom of SP tank
dz = 0.05; % z(20) is the injection level; z(21) is the 1st level after injection
for i = 1:1:32
    z(i) = (i - 1) * dz;
end
% Initial conditions calculated from the experiment
% Case 081804_lows_0air_50C
% Steam flow rate 0.07 kg/s, pure steam, initial pool T = 50 C
% DW pressure 230 kPa
m_s = 0.07; % Steam flow rate 0.07 kg/s
h_fg = 2713.12 - 525.154; % Latent heat of the steam under P = 230 kPa
hf_sat = 523.735; % Enthalpy for the saturated water under P = 230 kPa
T_sat = 125.019; % T_sat under P = 230 kPa
rouf_sat = 939.01; % Density for the saturated water under P = 230 kPa
Cp_f = 4.21657; % Averaged Cp_f from 50 C to T_sat under P = 230 kPa
T_f_ini = 51.5; % Steam gets contact with 51.5 C pool water
roup_ini = 988.09;
d_ini = 0.060634; % Injection port diameter calculated based on image analysis
thermal_k = 0.643654e-3; % Thermal conductivity in W/(m.C)
% Calculate the initial conditions for the injection port
% Using energy balance for steam and water system
% Assume that steam energy goes into the water to heat-up the water temperature
% Assume saturated water injection as results
% Energy equation:
% m_s * h_fg_sat = m_s * hf_sat + m_f * Cp_f * (T_sat - T_f_ini)
m_f = m_s * h_fg / (Cp_f * (T_sat - T_f_ini));
% Initialize the jet cell thickness h(?)
for i = 1:1:100
    h(i) = 0.0;
u(i) = 0.0;
d(i) = 0.0;
Tf(i) = 0.0;
rouf(i) = 0.0;
roup(i) = 988.09;
m(i) = 0.0;
dm(i) = 0.0;
E(i) = 0.0;

end
% \( m_s + m_f = r_ouf_sat \times A_{ini} \times u_0 \)
u(1) = (m_s + m_f)/(rouf_sat \times DiatoArea(d_{ini}));
d(1) = d_{ini};
Tf(1) = T_sat;
rouf(1) = rouf_sat;
h(1) = u(1) \times dt;
m(1) = rouf(1) \times DiatoArea(d(1)) \times h(1);
%Follow jet flow until it reaches to the water surface, check time requirement
k = 1;
flag_level = 1;  % flag indicates whether the jet reaches the surface of water
% flag_level = 1 does not reach surface
while(flag_level ~= 0)
    k = k + 1;
    % Calculate the entrainment flow rate
    % \( E_k = \pi \times Jet_d \times z_k \times rouf_k \times Ce \times uf_k \times dt \)
    Ce = 0.116;
    E(k) = \pi \times d(k - 1) \times h(k - 1) \times roup(k - 1) \times Ce \times u(k -1) \times dt;
    m(k) = m(k - 1) + E(k);
    Tf(k) = \left( m(k - 1) \times Tf(k - 1) + E(k) \times Tf_{ini} \right) / m(k);
    rouf(k) = Tf_to_rou_f(Tf(k));
    u(k) = \left( m(k - 1) \times u(k - 1) + m(k) \times \left( \frac{rouf(k) - rouf(k))}{rouf(k)} \right) \times g \times dt \right) / m(k);
    h(k) = u(k) \times h(k - 1) / u(k - 1);
    d(k) = \left( m(k)/(rouf(k) \times PI \times h(k)) \right) \times 0.5;
    % Check whether the jet reaches the surface of the water
    if(sum(h) < zs - z0)
        flag_level = 1;
    else
        flag_level = 0;
    end
end
% Calculate surface jet spreading time from jet
% Utilize similarity solution for the plane jet
% Jet centerline velocity proportional to x^(-1/3)
% \( Uc = \left(\frac{3M^2}{32\rho u^2 \mu}\right)^{0.5} x^{-1/3} \)
Mf = rouf(k) * u(k) * h(k);
miu = 546.87e-6/rouf(k);
A = \left(\frac{3Mf^2}{32rouf(k)^2muf}\right)^{0.5};
L = (A/u(k))^{1/3};
Time_surf = 0.75 * ((L+D_pool/2)^4/3 - L^4/3)/A;
% Calculate total energy transferred into the surface layer
Mass_t = m(k) * Time_surf / dt;
Temp_f = Tf(k);
% Calculate the averaged pool side surface layer temperature
Area_surf = DiatoArea(eqD) - DiatoArea(d(k));
h_surf = h(k);
V_surf = Area_surf * h_surf;
roup_surf = roup_ini;
m_surf = roup_surf * V_surf;
T_surf = (m_surf * Tf_ini + Mass_t * Temp_f) / (m_surf + Mass_t);
T_ini = T_surf + T_reach;
% From time T_reach + Time_surf to end of the test
% Need to consider the pool side energy influence
% Apply Euler Method to the Jet and Pool
% Initialize the velocity (u), temperature (T), density (rho) and diameter (d) for both pool side and jet side
% Use s(k) to store the hot jet path
s(1) = z0 + h(1);
for i = 2:1:Step_reach
    s(i) = s(i-1) + h(i);
end
% Use the format X(z,t), where z indicates the height, t indicates the time
% Initiate the jet parameters
t(1) = T_ini;
for i = 1:1:19
    roufj(i, 1) = roup_ini;  % jet water density
    Tempj(i, 1) = Tf_ini;  % jet water temperature
    ufj(i, 1) = 0.0;
    dfj(i, 1) = 0.0;  % jet diameter
    dfp(i, 1) = eqD;
    Vfp(i, 1) = \pi * dfp(i, 1) * dfp(i, 1) * dz / 4;
    Mfp(i, 1) = roufj(i, 1) * Vfp(i, 1);
    Afp(i, 1) = DiatoArea(dfp(i, 1));
end
for i = 20:1:32
    level = (i - 1) * dz;
l_flag = 1;
for k = 1:1:Step_reach-1
    if ((level < s(k+1) & (l_flag == 1)))
        ufj(i, 1) = u(k);
        Tempj(i, 1) = Tf(k);
        Tempp(i, 1) = Tf_ini;
        dfj(i, 1) = d(k);
        dfp(i, 1) = (eqD*eqD - dfj(i,1)*dfj(i,1)) ^ 0.5;
        Afp(i, 1) = DiatoArea(dfp(i, 1));
        roufj(i, 1) = rouf(k);
        roufp(i, 1) = roup_ini;
        Vfj(i, 1) = PI * dfj(i, 1) * dfj(i, 1) * dz / 4;
        Mfj(i, 1) = roufj(i, 1) * Vfj(i, 1);
        Afj(i, 1) = DiatoArea(dfj(i, 1));
        l_flag = 0;
    end
end
dfj(19, 1) = d_ini;
Afj(19, 1) = DiatoArea(dfj(19, 1));
ufj(19, 1) = u(1);
roufj(19, 1) = rouf_sat;
Tempj(19,1) = T_sat;
Mfp(32,1) = Mfp(31,1);
Mfj(32,1) = Mfj(31,1);
Afp(32,1) = Afp(31,1);
Tempp(32, 1) = T_surf;
roufp(32, 1) = Tf_to_rou_f(T_surf);
%Change dt to 0.5 second for transient calculation
dt = 0.35;
for t_step = 1:1:10000
t = dt * i;
dfj(19, 2) = d_ini;
Afj(19, 2) = Afj(19,1);
ufj(19, 2) = ufj(19,1);
roufj(19, 2) = roufj(19,1);
Tempj(19, 2) = Tempj(19,1);
Mfp(19, 2) = Mfp(19,1);
for i = 20:1:32
    %Calculate the entrainment flow in rate
    if (i < 32)
        E(i, 1) = PI * dfj(i, 1) * dz * roufp(i, 1) * Ce * ufj(i, 1);
        %Use mass equation to calculate the mass in the next time step
    end
end
% Use energy equation to calculate the jet temperature in the next time step
Tempj(i, 2) = (dt*(roufj(i-1,1)*ufj(i-1,1)*Afj(i-1,1)*Tempj(i-1,1) + E(i,1)*Tempp(i,1) - roufj(i,1)*ufj(i,1)*Afj(i,1)*Tempj(i,1)) + Mfj(i, 1)*Tempj(i,1)) / Mfj(i, 2);

% Calculate the jet fluid density at the next time step
roufj(i, 2) = Tf_to_rou_f(Tempj(i, 2));

% Calculate the jet diameter and jet area based on Mass and Density
Vfj(i,2) = Mfj(i,2)/roufj(i,2);
Afj(i,2) = Vfj(i,2)/dz;
dfj(i,2) = (4*Afj(i,2)/PI)^0.5;

% Calculate the pool diameter and pool area
dfp(i,2) = (eqD*eqD - dfj(i,2)*dfj(i,2))^0.5;
Afp(i,2) = PI * dfp(i,2) * dfp(i,2) / 4;
Vfp(i,2) = Afp(i,2) * dz;

% Use momentum equation to calculate the jet fluid velocity at the next time step
% Momentum convection term
M_conv = (roufj(i,1)*ufj(i,1)*ufj(i,1)*Afj(i,1) - roufj(i-1,1)*ufj(i-1,1)*ufj(i-1,1)*Afj(i-1,1))*dt/dz;

% Buoyancy force term
Buoy = (roufp(i,1) - roufj(i,1))*Vfj(i,1)*g*dt;

% Momentum equation
ufj(i, 2) = (Buoy - M_conv + roufj(i,1)*ufj(i,1)) / roufj(i,2);

% Use energy equation to calculate the pool temperature in the next time step
E_cond = dt*thermal_k*((Tempp(i+1,1) - Tempp(i,1))*Afp(i+1,1) - (Tempp(i,1) - Tempp(i-1,1))*Afp(i,1))/dz;

SumE = 0.0;
for k = 20:1:i
    SumE = SumE + E(k, 1);
end
if (i > 20)
    SumEk = SumE - E(i, 1);
else
    SumEk = 0.0;
end
Tempp(i, 2) = ((Mfp(i,1) - (E(i,1) + SumEk)*dt)*Tempp(i,1) + SumE*dt*Tempp(i+1,1) + E_cond) / Mfp(i,1);
if (Tempp(i, 2) >= 124)
    Tempp(i, 2) = 124.0;
end
roufp(i,2) = Tf_to_rou_f(Tempp(i, 2));
Mfp(i,2) = roufp(i, 2) * Vfp(i, 2);

else
  
  for cell 32 we only need the energy equation
  
  Tempj(i, 2) = Tempj(i-1, 2);
  Vfj(i, 2) = Vfj(i-1, 2);
  Afj(i, 2) = Afj(i-1, 2);
  dfj(i, 2) = dfj(i-1, 2);
  dfp(i, 2) = dfp(i-1, 2);
  Afp(i, 2) = Afp(i-1, 2);
  ufj(i, 2) = ufj(i-1, 2);
  SumEk = 0.0;
  for k = 20:1:i-1
    SumEk = SumEk + E(k, 1);
  end
  
  SumEk = SumE - E(i, t_step)
  E_cond = -dt*thermal_k*((Tempp(i,1) - Tempp(i-1,1))*Afp(i,1))/dz;
  Tempp(i, 2) = ((Mfp(i,1) - SumEk*dt)*Tempp(i,1) + dt*roufj(i-1,1)*Afj(i-1,1)*ufj(i-1,1)*Tempj(i-1,1) + E_cond ) / Mfp(i,1);
  roufj(i, 2) = Tf_to_rou_f(Tempj(i, 2));
  
  if (Tempp(i, 2) >= 124)
    Tempp(i, 2) = 124.0;
  end
  roufp(i,2) = Tf_to_rou_f(Tempp(i, 2));
  Vfp(i,2) = dz * DiatoArea(eqD);
  Mfp(i,2) = roufp(i, 2) * Vfp(i, 2);
  
end

end

%Calculate the thermal conductivity effect on pool water temperature
for i = 1:1:19
  if(i == 1)
    E_cond = dt*thermal_k*((Tempp(i+1,1) - Tempp(i,1))*Afp(i+1,1))/dz;
    Tempp(i, 2) = (Mfp(i,1)*Tempp(i,1) + E_cond) / Mfp(i,1);
  else
    E_cond = dt*thermal_k*((Tempp(i+1,1) - Tempp(i,1))*Afp(i+1,1) - (Tempp(i,1) - Tempp(i-1,1))*Af(i,1))/dz;
    Tempp(i, 2) = (Mfp(i,1)*Tempp(i,1) + E_cond) / Mfp(i,1);
  end
  
end

if (t_step / 10 == floor(t_step/10))
  re_time(t_step/10) = t;
  for i=1:1:32
    poolTemp(i,t_step/10) = Tempp(i,2);
  end
else
    for n=20:1:32
        Mfj(n,1) = Mfj(n,2);
        Tempj(n,1) = Tempj(n,2);
        roufj(n,1) = roufj(n,2);
        Vfj(n,1) = Vfj(n,2);
        Afj(n,1) = Afj(n,2);
        dfj(n,1) = dfj(n,2);
        dfp(n,1) = dfp(n,2);
        Afp(n,1) = Afp(n,2);
        ufj(n,1) = ufj(n,2);
        Tempp(n,1) = Tempp(n,2);
        roufp(n,1) = roufp(n,2);
        Mfp(n,1) = Mfp(n,2);
    end
end
end

figure(1);
plot(z, poolTemp(:,1000), 'k');
grid on;
hold on;
plot(z, poolTemp(:, 840), 'r');
plot(z, poolTemp(:, 550), 'g');
plot(z, poolTemp(:, 270), 'b');
plot(z, poolTemp(:, 130), 'c');
TestData = [1.54 57.6265 61.8955 71.75 82.752 88.4729;
            1.486 56.8252 63.5518 75.073 84.49 88.4981;
            1.459 58.7092 64.2989 74.2946 82.5558 87.2331;
            1.23 57.7194 63.9155 73.4644 82.286 86.3565;
            0.989 53.5221 57.4031 66.8142 76.8965 82.5072;
            0.76 51.4182 51.5881 51.6147 51.5988 51.6424;
            0.455 51.4043 51.2680 51.4253 51.297 51.3324;
            0.15 51.0976 51.2041 51.1769 51.062 51.0668;
            ];
hold on;
plot(TestData(:,1), TestData(:,6), ':ok');
plot(TestData(:,1), TestData(:,5), ':xr');
plot(TestData(:,1), TestData(:,4), ':sg');
plot(TestData(:,1), TestData(:,3), ':db');
plot(TestData(:,1), TestData(:,2), ':vc');
legend('3500 Sec', '3000 Sec', '2000 Sec', '1000 Sec', '500 Sec');
xlabel('z (m)');
ylabel('Temperature (^0C)');
Appendix C User-defined Function in CFD Calculation

/* UDF for SP mixing */
#include "udf.h"
DEFINE_PROPERTY(liq_rou, c, tf)
/* water density under 2.3 bar; Temperature ranges from 40 to 124.6 degree */
real rou_f;
real Temp_f = C_T(c, tf);
real a0 = 1000.0;
real a1 = -0.0043;
real a2 = -0.0074;
real a3 = 5.0e-5;
real a4 = -2.0e-7;
real a5 = -4.0e-10;
Temp_f = Temp_f - 273.15;
if( (Temp_f >= 40.0) && (Temp_f <= 124.6) ){
    rou_f = a0 + a1*Temp_f + a2*Temp_f*Temp_f;
    rou_f = rou_f + a3*pow(Temp_f, 3.0) + a4*pow(Temp_f, 4.0);
    rou_f = rou_f + a5*pow(Temp_f, 5.0);
}
else if(Temp_f > 124.6){
    rou_f = 939.28;
}
else{
    rou_f = 992.3;
}
return rou_f;
}
DEFINE_MASS_TRANSFER(cond_rate, cell, thread, from_index, from_s_index, to_index, to_s_index){
real m_fg, T_liq, T_vapor, alpha_vapor, htc, i_fg, d_0;
real T_sat = 397.8;
Thread *tg = THREAD_SUB_THREAD(thread, from_index);
Thread *tf = THREAD_SUB_THREAD(thread, to_index);
htc = 2500.0; /* heat transfer coefficient, W/m2-K */
i_fg = 2188.92E+03; /* latent heat, J/kg */
d_0 = 3.0E-03; /* typical vapor bubble size, m */
m_fg = 0.0;
alpha_vapor = C_VOF(cell, tg);
T_liq = C_T(cell, tf);
T_vapor = C_T(cell, tg);

if(T_liq >= T_sat){
    m_fg = -0.05 * C_VOF(cell, tf) * C_R(cell, tf) * fabs(T_liq - T_sat) / T_sat;
} else{
    if( alpha_vapor >= 0.01 ){ /* vapor exists */
        m_fg = alpha_vapor * htc * 6.0 * fabs(T_sat - T_liq) / d_0 / i_fg;
    } else{
        m_fg = 0.0;
    }
}
return m_fg;

VITA
VITA

Ling Cheng

EDUCATION

- Doctor of Philosophy, August 2015
  School of Nuclear Engineering
  Purdue University

- Master of Science, July 2000
  Institute of Nuclear Energy and Technology
  Tsinghua University