MODELING OF DISPERSED FLOW FILM BOILING WITH TWO FLOW, FIVE FIELD EULERIAN- EULERIAN APPROACH AND EFFECTS OF SPACER GRIDS ON HEAT TRANSFER

A Thesis in

Nuclear Engineering

by

Sule Ergun

© 2006 Sule Ergun

Submitted in Partial Fulfillment of the Requirements for the Degree of

Doctor of Philosophy

May 2006
The thesis of Sule Ergun was reviewed and approved* by the following:

Lawrence E. Hochreiter  
Professor of Mechanical and Nuclear Engineering  
Thesis Co-Advisor  
Co-Chair of Committee

John H. Mahaffy  
Professor of Nuclear Engineering  
Thesis Co-Advisor  
Co-Chair of Committee

Fan-Bill Cheung  
Professor of Mechanical and Nuclear Engineering

Kostadin Ivanov  
Professor of Nuclear Engineering

Savas Yavuzkurt  
Professor of Mechanical Engineering

Cengiz Camci  
Professor of Aerospace Engineering

Jack Brenizer  
Professor of Mechanical and Nuclear Engineering  
Head of the Department of Nuclear Engineering Program

*Signatures are on file in the Graduate School
ABSTRACT

In the case of a postulated loss of coolant accident (LOCA) in a nuclear reactor, an accurate prediction of clad temperature is needed to determine the safety margins. The large break LOCA analyses can be divided into three time periods. These periods are blowdown, refill and reflood. During the blowdown and reflood phases of the LOCA, when the local void fraction is greater than 80% and the wall is at a temperature above minimum film boiling temperature ($T_{\text{min}}$), heat is transferred from the fuel rod to a continuous vapor flow with dispersed droplets. The high void fraction mixture of droplets and vapor provide cooling to prevent the clad temperature from exceeding the safety limit. The heat transfer process for high void fraction mixture is called dispersed flow film boiling (DFFB). This thesis has been modeled DFFB in the reflood phase of a LOCA in a pressurized water reactor (PWR) rod bundle.

In this study, the modifications and modification requirements for the COBRA-TF code to obtain a five field Eulerian - Eulerian modeling for two-phase DFFB is described. COBRA-TF is a best estimate code developed for the rod bundle analysis and has four fields, namely, vapor, entrained drop and continuous liquid film. COBRA-TF has a detailed reflood package which takes effect of spacer grids on heat transfer into account. This study has a detailed description of code’s solution scheme and the models used for dispersed flow film boiling.

The dispersed flow film boiling heat transfer model of the COBRA-TF code has been modified by adding a small droplet field to the code as the fifth field. The effect of smaller, thermally more active droplets on heat, mass and momentum transfer during DFFB has been modeled.

Since the large drop break up due to spacer grids is one of the reasons for small droplet generation, the spacer grid models of the COBRA-TF have been revised and modified. In addition to small droplet generation, the spacer grid rewet is an important aspect of heat transfer during DFFB. Since wet spacer grids provide a large interfacial
area for the heat transfer between the superheated vapor and the liquid deposited on the spacer grid, the grid rewet has been modeled and effect of wet grid on heat transfer has been investigated.

Adding a new field to the code not only requires adding new equations and models to the code but also makes changes in existing equations and models necessary. The changes in currently existing field equations and closure relations such as entrainment model have been described in this study.

Once the code modifications are performed, the code evaluation with proper experimental data has been presented. The rod bundle reflood experiments have been selected, described and code modeling for these experiments have been introduced. Reflood experiments during which DFFB exists have been selected from Full Length Emergency Core Heat Transfer-System Effects and Separate Effects Tests (FLECHT-SEASET) and Rod Bundle Heat Transfer (RBHT) experiments for the evaluation.

The results of the code evaluations have been presented by comparing the experimental data with the results of code simulations performed with original and modified code. Measurements and calculations for the heater rod, vapor and grid temperatures and quench front progression have been compared.

With the new smaller droplet field, the rod to flow heat transfer, the vapor velocity, interfacial heat and mass transfer, droplet mass and volume distribution, i.e., the thermal and mechanical non-equilibrium between the continuous vapor and dispersed droplet phases has been modeled accurately. This is an important improvement over the dispersed flow film boiling model of the original code. The results of the analysis performed with the modified code have been indicated improvement in code predictions for the rod surface temperature, vapor temperature and quench front behavior.
# TABLE OF CONTENTS

LIST OF FIGURES ........................................................................................................ viii

LIST OF TABLES ........................................................................................................... xi

ACKNOWLEDGEMENTS ............................................................................................... xii

Chapter 1  INTRODUCTION ...................................................................................... 1

1.1 Dispersed Flow Film Boiling ............................................................................. 4
1.2 Effects of Spacer Grids on Heat Transfer ....................................................... 11
1.3 Two Phase Flow Modeling with Eulerian Vapor and Liquid Fields .......... 16

Chapter 2  LITERATURE REVIEW ........................................................................... 19

2.1 Experimental Studies and Models Developed for DFFB ............................... 19
2.4 Comprehensive Models for DFFB................................................................. 30
2.5 Summary of the Literature Survey and Purpose of the Present Study ........... 33

Chapter 3  CURRENT APPROACH TO MODEL DFFB AND SPACER GRID EFFECTS IN COBRA-TF ................................................................................. 35

3.1 Multiphase Flow Modeling ............................................................................. 35
  3.1.1 Exact Integral Equations for Two Phase Flow ..................................... 36
  3.1.2 Differential Single Phase Equations ..................................................... 39
  3.1.3 Time Average Equations ...................................................................... 42
3.2 COBRA-TF Two Fluid, Three Field Equations ............................................. 48
  3.2.1 Interfacial Exchange Terms .................................................................. 49
  3.2.2 COBRA-TF Assumptions for Three Field Modeling ............................ 51
  3.2.3 Conservation Equations ........................................................................ 52
3.3 COBRA-TF Numerical Solution Method....................................................... 55
  3.3.1 Finite Difference Equations .................................................................. 56
  3.3.2 Numerical Solution ............................................................................... 61
    3.3.2.1 Solution of Momentum Equations ............................................. 61
    3.3.2.2 Solution of Mass and Energy Equations .................................... 62
3.4 COBRA-TF Heat Transfer Package .............................................................. 64
3.5 Small Droplet Field ......................................................................................... 73
3.6 Spacer Grid Model in COBRA-TF ................................................................. 79
  3.6.1 Convective Enhancement ..................................................................... 79
  3.6.2 Grid Rewetting ..................................................................................... 80
  3.6.3 Droplet Breakup Model ........................................................................ 83
3.7 Interfacial Area Transport Equation ............................................................... 87

Chapter 4  PROPOSED MODEL AND DISCUSSION OF THE SOLUTION ........ 90
Appendix B  Quench Front Program ........................................................................... 206
LIST OF FIGURES

Figure 1-1: PWR Coolant Flow and Emergency Core Cooling System ................. 3
Figure 1-2: LOCA Transient Periods ..................................................................... 4
Figure 1-3: Flow Regimes with High and Low Reflood Rates [1] ......................... 6
Figure 1-4: Flow Regimes with Low Reflood Rate [1] ......................................... 7
Figure 1-5: Heat and Mass Transfer Mechanisms in DFFB .................................... 9
Figure 1-6: Mixing Vane Spacer Grids used in Penn State University RBHT test Facility [11] ............................................................................................................. 12
Figure 1-7: The Effects of Spacer Grids on Heat Transfer .................................... 13
Figure 1-8: ERSEC data for heater rod temperature rise at each elevation for different grid types ............................................................... 15
Figure 1-9: Rod Surface and Vapor Temperature Profile from RBHT Data............ 16
Figure 3-1: The Two Phase Volume used in Derivations ..................................... 37
Figure 3-2: COBRA-TF Mesh Cells [60] ............................................................... 55
Figure 3-3: Boiling Curve used in COBRA-TF ..................................................... 67
Figure 3-4: Heat Transfer Selection used in COBRA-TF ...................................... 67
Figure 3-5: Radiation Heat Flux Network used for Radiant Heat Flux to the Grid [60] ................................................................................................................. 82
Figure 3-6: Droplet Slicing due to Spacer Grid ..................................................... 84
Figure 4-1: Results of Kataoka et al. Correlation and the Curve Fit to the Results..... 106
Figure 4-2: Bag Type (Aerodynamic) Break up [45] ............................................. 107
Figure 4-3: Variation of maximum residual droplet diameter with Weber number ... 110
Figure 4-4: Shear (Capillary) Break up [45] ......................................................... 111
Figure 4-5: Grid Temperature Measurement, RBHT Experiment 1096 (20 psia, 1.0 in./sec Inlet Flooding Rate, 20 ºF Inlet Subcooling) ................................. 122
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>4-6</td>
<td>Comparison of the Grid Temperature Measurement and the Grid Temperature Calculated with the Original COBRA-TF. (RBHT Experiment 1096 (20 psia, 1.0 in./sec Inlet Flooding Rate, 20 °F Inlet Subcooling)) .......... 123</td>
</tr>
<tr>
<td>4-7</td>
<td>Convective Heat Transfer Enhancement Due to Droplets ....................... 130</td>
</tr>
<tr>
<td>5-1</td>
<td>FLECHT SEASET Bundle Cross Section [42]........................................ 135</td>
</tr>
<tr>
<td>5-2</td>
<td>Schematic of FLECHT-SEASET Test Loop [42]................................. 136</td>
</tr>
<tr>
<td>5-3</td>
<td>RBHT Test Schematic [77]......................................................... 139</td>
</tr>
<tr>
<td>5-4</td>
<td>Rod Bundle Cross Section View.................................................. 141</td>
</tr>
<tr>
<td>6-1</td>
<td>Partitioning of Channels and Sections ...................................... 148</td>
</tr>
<tr>
<td>6-2</td>
<td>RBHT Heated Rod Materials and Nodalization .................................. 149</td>
</tr>
<tr>
<td>6-3</td>
<td>RBHT Unheated Rod and Housing Nodalization................................. 149</td>
</tr>
<tr>
<td>6-4</td>
<td>RBHT Rod Bundle Axial Power Distribution ..................................... 150</td>
</tr>
<tr>
<td>6-5</td>
<td>Heater Rod Temperature at 2.69 m (106.0 in.), Experiment 1383 ........ 152</td>
</tr>
<tr>
<td>6-6</td>
<td>Steam Temperature at 2.54 m (100.0 in.), Experiment 1383 .............. 153</td>
</tr>
<tr>
<td>6-7</td>
<td>Heater Rod Temperature at 1.9 m (74.9 in.), Experiment 1383 .......... 154</td>
</tr>
<tr>
<td>6-8</td>
<td>Steam Temperature at 1.9 m (74.9 in.), Experiment 1383 ............... 155</td>
</tr>
<tr>
<td>6-9</td>
<td>Heater Rod Temperature at 2.79 m (110.0 in.), Experiment 1383 ....... 156</td>
</tr>
<tr>
<td>6-10</td>
<td>Heater Rod Temperatures above and below the 6th Spacer Grid - Data, Experiment 1383 ................................................................. 157</td>
</tr>
<tr>
<td>6-11</td>
<td>Heater Rod Temperatures above and below the 6th Spacer Grid – Modified Code, Experiment 1383 ................................................................. 158</td>
</tr>
<tr>
<td>6-12</td>
<td>Grid Temperature, Grid 6, Experiment 1383 .................................... 159</td>
</tr>
<tr>
<td>6-13</td>
<td>Quench Front Location, Experiment 1383 ..................................... 161</td>
</tr>
<tr>
<td>6-14</td>
<td>Liquid Mass Stored in the Bundle, Experiment 1383 ....................... 162</td>
</tr>
<tr>
<td>6-15</td>
<td>Heater Rod Temperature at 2.69 m (106.0 in.), Experiment 1096 ........ 164</td>
</tr>
<tr>
<td>6-16</td>
<td>Steam Temperature at 2.54 m (100.0 in.), Experiment 1096 ............. 165</td>
</tr>
</tbody>
</table>
Figure 6-17: Heater Rod Temperature at 1.9 m (74.9 in.), Experiment 1096 .......... 166
Figure 6-18: Steam Temperature at 1.9 m (74.9 in.), Experiment 1096 ............... 167
Figure 6-19: Heater Rod Temperature at 2.79 m (110.0 in.), Experiment 1096 ....... 169
Figure 6-20: Heater Rod Temperatures above and below the 6th Spacer Grid – Data, Experiment 1096 ................................................................. 170
Figure 6-21: Heater Rod Temperatures above and below the 6th Spacer Grid – Modified Code, Experiment 1096 ......................................................... 171
Figure 6-22: Grid Temperature, Grid 6, Experiment 1096 .................................... 172
Figure 6-23: Quench Front Location, Experiment 1096...................................... 173
Figure 6-24: Liquid Mass Stored in the Bundle, Experiment 1096...................... 174
Figure 6-25: Heater Rod Temperature at 2.28 m (90.0 in.), Experiment 31504 .... 177
Figure 6-26: Heater Rod Temperature at 2.13 m (84.0 in.), Experiment 31504 .... 178
Figure 6-27: Steam Temperature at 2.44 m (96.0 in.), Experiment 31504 ............ 179
Figure 6-28: Steam Temperature at 3.05 m (120.0 in.), Experiment 31504 .......... 180
Figure 6-29: Quench Front Location, Experiment 31504................................. 181
Figure 6-30: Heater Rod Temperature at 2.28 m (90.0 in.), Experiment 31922 .... 183
Figure 6-31: Heater Rod Temperature at 2.13 m (84.0 in.), Experiment 31922 .... 184
Figure 6-32: Steam Temperature at 2.44 m (96.0 in.), Experiment 31922 ............ 185
Figure 6-33: Steam Temperature at 3.05 m (120.0 in.), Experiment 31922 .......... 186
Figure 6-34: Quench Front Location, Experiment 31922................................. 187
Figure 6-35: Heater Rod Temperature at 2.69 m (106.0 in.), Experiment 1096, Comparison of Data, Modified Code Calculation and Modified Code Calculation with Altered Interfacial Drag Coefficient ......................... 188
<table>
<thead>
<tr>
<th>Table 3-1:</th>
<th>Time Averaged Two Phase Flow Variables</th>
<th>44</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table 3-2:</td>
<td>The Surface Averaged Variables on the Phase Interface</td>
<td>45</td>
</tr>
<tr>
<td>Table 4-1:</td>
<td>Equations for the New COBRA-TF Solution Scheme</td>
<td>92</td>
</tr>
<tr>
<td>Table 4-2:</td>
<td>Modifications to Existing COBRA-TF Conservation Equations</td>
<td>94</td>
</tr>
<tr>
<td>Table 5-1:</td>
<td>Conditions for FLECHT-SEASET Run 31504</td>
<td>137</td>
</tr>
<tr>
<td>Table 5-2:</td>
<td>Conditions for FLECHT-SEASET Run 31922</td>
<td>137</td>
</tr>
<tr>
<td>Table 5-3:</td>
<td>Conditions for RBHT Run 1383</td>
<td>143</td>
</tr>
<tr>
<td>Table 5-4:</td>
<td>Conditions for RBHT Run 1096</td>
<td>143</td>
</tr>
</tbody>
</table>
ACKNOWLEDGEMENTS

I would like to thank my thesis advisors Dr. Lawrence E. Hochreiter and Dr. John H. Mahaffy for their support, guidance and advice for my thesis research.

I would also thank to my other committee members for their contributions: Dr. Fan-Bill Cheung, Dr. Cengiz Camci, Dr. Kostadin Ivanov and especially Dr. Savas Yavuzkurt for the enlightening discussions we had throughout the entire time.

The financial support of Nuclear Regulatory Commission for the Rod Bundle Heat Transfer project is also acknowledged. Without the reflood experimental data, this work would not be a novel one.

I would like to acknowledge the support from my former office mates Dr. Ruwan K. Ratnayake, Dr. Arunkumar Sridharan, Dr. Jong E. Chang and Dr. Mark Holowach. The discussions we had for this thesis work were truly very useful and enlightening.

For her support and encouragement I would like to thank to my best friend Dr. Aylin Yilmazbayhan. I also would like to thank Ahmet Turhan for his friendship and support during the most difficult times of my PhD experience.

Last but not least I would like to thank my parents, Erbay and Sohret Ergun for their sacrifices and support. I dedicate this thesis work to them. Without my parents and my sisters (Gozde, Hale and Ece), this work would not have been possible.
Chapter 1

INTRODUCTION

The current study aims to model dispersed flow film boiling which is one of the heat transfer regimes occurring during reflood phase of a loss of coolant accident. The purpose of the modeling is to improve the predictive capability of currently existing COBRA-TF code which has an Eulerian-Eulerian four-field solution scheme for the two-phase flow calculations. The improved modeling includes adding a separate smaller droplet field to the code’s solution scheme and improving models used to calculate effects of the spacer grids on the heat transfer.

In the case of a postulated loss of coolant accident (LOCA) in a nuclear reactor, the prediction of clad temperature is very important to determine the safety margins. The postulated LOCA is the worst accident that can occur in the primary system of a Pressurized Water Reactor (PWR). The primary system of a PWR is illustrated in Figure 1-1. The worst case large break LOCA is postulated to occur as a result of double-ended large break in one of the cold legs. As a result of Nuclear Regulatory Commission’s Appendix K requirements, in order to test the performance of the Emergency Core Cooling System (ECCS) of the reactor, analyses are performed to simulate LOCA.

The analyses, performed for this LOCA, group the accident sequence in three periods. These periods are blowdown, refill and reflood. Figure 1-2 presents the accident sequence and time spanned during these periods.

During blowdown, following a large break in the cold leg, subcooled reactor coolant is discharged to the reactor containment. Because of the large pressure difference between the primary system and the reactor containment, the coolant is discharged very fast. Due to the discharge of the coolant, the fuel rods dry and their temperatures increase.
The emergency coolant is injected to the system early in the blowdown and rod cooling is supplied by the two phase mixture.

The refill period of LOCA starts when the accumulators passively initiate injecting subcooled coolant to the system. Analysis shows that, due to counter current steam velocities, at the beginning of refill, the coolant from the accumulators do not penetrate to the lower plenum, instead it is pushed out from the cold leg through bypass. Once countercurrent steam velocity decreases, water penetrates to the lower plenum and fills it. Refill lasts for about 10.0 seconds and it ends once the water level in lower plenum reaches to the bottom of the fuel rods.

The reflood period lasts several minutes. During reflood, the emergency coolant fills the vessel and quenches the fuel rods. The quenching occurs once the very high fuel rod temperature drops below the minimum boiling temperature and the liquid wets the fuel rod surface. The quenching causes the transfer of large amount of heat from fuel rods to coolant and causes steam formation. The steam entrains drops and carries them upward with it. Entrained drops help to cool the steam and therefore the fuel rod at higher elevations. Once the reflood period of LOCA is over, a long term cooling is needed to transfer decay heat from fuel rods.
Figure 1-1: PWR Coolant Flow and Emergency Core Cooling System
1.1 Dispersed Flow Film Boiling

During the blowdown and reflood phases of the LOCA, when the local void fraction is greater than 80% and the wall is at a temperature above minimum film boiling temperature ($T_{\text{min}}$), the heat is transferred from rod to a continuous vapor flow with dispersed droplets. The high void fraction mixture of droplets and vapor provide the cooling to prevent clad temperature from exceeding safety limit and must be modeled.
accurately. This high void fraction mixture is called dispersed or mist flow and heat transfer mode which occurs when the flow is dispersed flow is referred as dispersed flow film boiling (DFFB). Several researchers have performed studies to investigate and model DFFB in blowdown and reflood phases. The current study aims to model DFFB for the reflood phase of a LOCA in a rod bundle.

The DFFB phenomena change with the conditions initiating them. Depending on the velocity of the quench front (inlet flooding rate) and the quality at the quench front, two flow regimes are observed during the reflooding. Figure 1-3 [1] presents these two cases. When flooding rate is high (~0.15 m/sec - ~6 in/sec-) the dominant regime above the quench front is inverted annular regime. In this regime, the rod is covered with a thin layer of vapor and rest of the channel is filled with subcooled liquid. Once the liquid reaches the saturation, it breaks into liquid chunks and slugs; dispersed flow may or may not be observed later if more liquid break up occurs. Since the heat transfer in this regime is high, the clad temperature turns around earlier in the transient; therefore the peak clad temperature is lower in this regime.

When the inlet flow rate is lower (~0.0254 m/sec - ~1.0 in/sec-), the liquid reaches saturation quickly at or below the quench front and no subcooled inverted annular regime is observed. Between the quench front and dispersed flow, a transition regime with liquid chunks and slugs occurs. The droplets in the dispersed flow are created by the sputtering, bubble bursting and entrainment of liquid from the liquid filaments in this transition regime.
In the low inlet flooding cases, the dominant flow regime is DFFB during which the poor heat transfer occurs; therefore the peak clad temperature is observed. In DFFB, heat transfer occurs between the rod and the superheated vapor and droplet mixture. Since the droplet evaporation in the steam causes a decrease in the steam temperature and increase in the steam flow rate, the presence of the droplets in the steam helps to decrease...
the peak clad temperature [2]. Figure 1-4 [1] presents a detailed picture of the low flooding rate flow regimes.

Figure 1-4: Flow Regimes with Low Reflood Rate [1]
The heat transfer in DFFB has several components and none of them plays the major or dominant role. Figure 1-5 shows the heat and mass transfer mechanisms in DFFB. These mechanisms are summarized below.

- The heat transfer from heated surface to flow occurs basically by the convection to superheated vapor. At the axial locations far from the quench front, wall temperature is high enough to make droplet-wall contact impossible, so vapor becomes the only phase in contact with the heated surface. The data from FLECHT-SEASET experiment 31504 shows that the vapor superheat reaches up to 973 K [3].

- In addition to the convection heat is transferred from rods to flow by surface radiation to vapor and droplets. In modeling the heat transfer in DFFB, radiation heat transfer should also be emphasized. For the higher elevations at the rod surface, the temperature difference between rod surface and vapor, and rod surface and droplet can be high enough to make radiation heat transfer significant [4]. Low reflood rate data from FLECHT SEASET experiment 31504 show that, the difference in rod surface and vapor temperatures can be as high as about 573 K (~500 °F). The difference between rod surface-droplet temperatures can reach up to 973 K (~1300 °F) [3].

- Heat is transferred between droplets and superheated vapor through the interface of these phases. As mentioned earlier, the temperature difference between the vapor and liquid phases is high, this causes droplet evaporation. Besides the vapor superheat, for the heat transfer between the phases, the interfacial surface area is important. The distribution of droplets through the channel has an important effect on heat transfer. The droplets closer to the rod surface are more effective in decreasing vapor temperature and increasing wall to flow heat transfer [3], [5].

- For the elevations near quench front, due to the droplet-wall contact, heat is transferred from the rod to the droplets directly. Drop-wall contact has a minor role in heat transfer in DFFB, however, should be emphasized in a heat transfer
model for DFFB for the elevations at which the rod temperature is below minimum film boiling temperature [2], [6], [7].

- Droplets enhance the heat transfer to the vapor. The turbulence in the flow increases because of the interfacial shear between dispersed droplets and the continuous phase. The increase in turbulence enhances the convective heat transfer [6], [8].

- The structures (mainly the spacer grids) enhance heat transfer in the rod bundle due to flow acceleration, increased turbulence, rewetting and droplet breakup. The effects of grids are discussed in detail below.

Figure 1-5: Heat and Mass Transfer Mechanisms in DFFB
The most important characteristic of DFFB is the mechanical and thermal non-equilibrium between the continuous phase and dispersed phase. The thermal non-equilibrium between the phases results in the heat transfer from steam to droplets to evaporate them. This causes a decrease in steam temperature and an increase in convection heat transfer due to the increase in vapor flow and lower vapor temperature.

In addition to the temperature difference between the phases, heat transfer depends on the interfacial area between the vapor and droplets. The spectrum of droplet diameters, velocities, droplet concentration and droplet sizes are important parameters to determine the interfacial area between the entrained drops and superheated vapor.

Since the velocity difference between the phases affects the heat transfer, in order to model DFFB, the mechanical non-equilibrium must be emphasized. Droplet concentration at different elevations strongly depends on the velocity difference [2]. This indicates that, the coupling between the phases is very important. The heat transfer characteristics and the turbulence of vapor depends on the dispersed droplets and heat and mass transferred from droplet field and droplet field velocity and size distribution depend on vapor phase [9], [10].

The thermal and mechanical non-equilibrium between the phases is taken into account by the Nuclear Regulatory Commission’s best estimate codes. In order to simulate the characteristics of DFFB, closure relations for heat and momentum transfer mechanisms and droplet sizes and droplet diameter spectrum are used in these codes.
1.2 Effects of Spacer Grids on Heat Transfer

Spacer grids are the support structures used in nuclear fuel assemblies to space the fuel rods. Figure 1-6 [11] presents a drawing of the spacer grid used in Rod Bundle Heat Transfer (RBHT) experiment set up.

Spacer grids have several important effects on heat transfer in dispersed droplet film boiling regime [12]. The effects of spacer grids which cause an increase in the heat transfer are summarized below.

1) Single-Phase Heat Transfer Enhancement
   - Spacer grids cause disruption of fluid and thermal boundary layers on the fuel rod. Vapor flow accelerates within the grid then it expands downstream of the grid. This causes a disruption in both thermal and fluid boundary layers. Disruption and reestablishment of boundary layer increases the heat transfer. This increase in heat transfer downstream of the grid can be modeled with entrance effect phenomenon [13], [14], [15]. The disruption effect increases when spacer grids with mixing vanes are used.

   Some types of spacer grids have mixing vanes and flow tabs. Figure 1-6 shows a spacer grid with a mixing vane. Mixing vanes promote mixing downstream of the grid i.e. they cause an increase in the turbulent intensity [16]. In addition, they push droplets towards the fuel rods and make droplet-fuel rod contact easier. This also increases the heat transfer from rod surface [17].

   - The spacer grids are colder than the fuel rods they surround. Colder spacer grids are heated by radiation heat transfer from the rods; therefore they help cool the vapor and the fuel rods [3].
2) Two-Phase Heat Transfer Enhancement

- The spacer grids are unpowered structures and they quench earlier than the fuel rods. Once the spacer grid is wetted, it is covered with a liquid film. This creates an additional liquid-vapor interfacial area. Therefore it causes an increase in the liquid-vapor interfacial heat transfer which results in de-superheating of the vapor. In addition, the liquid film on the wet grid evaporates causing an increase in the vapor velocity, therefore an increase in convective heat transfer. These two mechanisms cause a decrease in vapor temperature downstream of the grids [3].

Figure 1-6: Mixing Vane Spacer Grids used in Penn State University RBHT test Facility [11]
• The grids can shatter the entrained droplets into smaller, thermally more active droplets. The small droplets easily evaporate and provide additional steam flow, therefore increase the convective heat transfer between the rods and the vapor [18], [19], [20].
• Spacer grids can cause de-entrainment since the droplets impinging on the wettable grid surface can form a liquid film. This liquid film can also be entrained into the flow. The entrained droplets increase convective heat transfer as mentioned earlier [21], [22].

The effects of the spacer grids on heat and mass transfer mechanisms are illustrated in Figure 1-7
Figure 1-8 presents the data taken from ERSEC experiment for the heater rod temperature rise at each elevation for different grid types. The locations of the spacer grids are shown in the figures. The same figure also shows that the drop in rod temperature downstream of the grid is up to 433 K (~300 °F) at upper elevations, when the inlet mass velocity is high.

The change in axial vapor and rod temperature due to spacer grids is presented in Figure 1-9. The data shown in the figure are taken from RBHT experiment 945. As the figure presents, the decrease in the rod surface temperature downstream of the grid is greater than 403 K (~200 °F) for the upper elevations.
Figure 1-8: ERSEC data for heater rod temperature rise at each elevation for different grid types
1.3 Two Phase Flow Modeling with Eulerian Vapor and Liquid Fields

The two phase flow is numerically modeled in several best estimate codes for the nuclear reactor safety analysis. These best estimate codes model the liquid and vapor phases as separate flows and use either two fields for the phases (liquid and vapor as in RELAP, TRAC series codes) or three fields (liquid, vapor and droplet fields as in COBRA-TF). In addition to these fields, non-condensable gases are also modeled as a separate field.

The best estimate codes solve the two phase flow problem using an Eulerian description for the fields. In the Eulerian description of the flow, the independent variables are a spatial point and time. In this description, the simulation is performed to
find out the changes in the flow at a given spatial point with time. The thermal and mechanical non equilibrium and the coupling of the phases are modeled using closure relationships that are empirical or semi-empirical.

The two phase flow model should have a separate droplet field to model DFFB in detail as it is modeled in COBRA-TF. Also, a model is needed to simulate the effects of spacer grids on the droplet field and the resulting interfacial heat transfer between the vapor field and the thermally more active small droplets. The objective of this thesis is to add the small droplet field to the two phase flow model of COBRA-TF as a separate field and therefore, to model the DFFB accurately.

The major contributions of this study to two phase flow modeling are:

- Adding an Eulerian small droplet field to the solution system and adding a separate interfacial area transport equation for this field, and therefore adding the detail of the droplet behavior, its interaction with spacer grids and its effects on rod heat transfer.
- Explicitly treating the effect of spacer grids on heat transfer in DFFB more accurately since the spacer grids are the major cause of the small droplet generation.

This thesis study utilizes a novel concept of adding a small droplet field to the two phase flow model in order to obtain a better simulation of interaction between the droplet fields and spacer grids.

Chapter 2 of the thesis is the review of the studies that have been performed to model DFFB and effects of spacer grids. The review summarizes the studies performed to develop heat, mass and momentum transfer and droplet hydrodynamics models, and the experiments performed to measure the quantities important for the modeling. Chapter 2 has also a short discussion on the existing best estimate codes modeling DFFB and spacer grids.
Chapter 3 has a brief discussion on COBRA-TF code and summarizes the conservation equations solved, the numerical methodology used to solve the equations and the models used as the closure relationships for the DFFB phase. The current spacer grid models existing in COBRA-TF are also described in Chapter 3. The purpose of this chapter is to introduce the solution system that is modified in this study and to put together the COBRA-TF equations and models for all COBRA-TF users.

Chapter 4 describes the models and model improvements that current study requires. The conservation equations added for the new smaller droplet field and the models required describing thermal and mechanical non equilibrium between vapor and new smaller droplet field are introduced in this chapter. The interfacial area transport equation for the smaller droplet field is also described. In addition, improvements to spacer grid models are presented.

The rod bundle experiments used to present the improvement in COBRA-TF predictions are described in Chapter 5. These experiments are the selected reflood tests from Full Length Emergency Core Heat Transfer-System Effects and Separate Effects Tests (FLECHT SEASET) and Rod Bundle Heat Transfer (RBHT) experiments mentioned in Chapter 5.

Chapter 6 includes the results of the code simulations. The results include the comparison of the experimental data and the code’s predictions for the important quantities to describe DFFB phenomena.

Chapter 7 summarizes the improvements of the present work and conclusions. The recommendations for the future work are also given in this chapter.
Chapter 2

LITERATURE REVIEW

In this chapter, the models evaluated for phenomena related with DFFB and related data are summarized. In addition, a survey of these models and data are presented to demonstrate the need for the current work. The data from reflood experiments performed for tubes and fuel rod bundles are identified in order to provide a basis for the model development and evaluation. The rod bundle experiments and data obtained during these experiments are especially investigated to present the effects of spacer grids on the heat transfer during DFFB.

Since the current work is performed to develop an accurate model for DFFB and therefore a better simulation of droplet effects on heat transfer, a literature survey for experimental and theoretical studies on droplet hydrodynamics is summarized. At the end of the literature survey, comprehensive numerical and mechanistic models simulating DFFB and effects of spacer grids are presented. As a conclusion to this chapter, a discussion on current models for DFFB and the need for the current study is presented.

2.1 Experimental Studies and Models Developed for DFFB

There has been significant research and studies performed to model DFFB including experimental studies to observe and measure the characteristics of DFFB. Using the data from the experiments, empirical or semi empirical models and correlations have been developed for the phenomena related with the DFFB such as heat and mass transfer.

There are several experiments that supply data for model development and testing. These experiments were performed either for tube or bundle geometries. Since
this study aims to model DFFB in rod bundles, the data from rod bundle tests are investigated for the model evaluation and assessment. Full Length Emergency Core Heat Transfer-System Effects and Separate Effects Tests (FLECHT-SEASET) [3] are one of these bundle tests and were performed by Westinghouse for a typical PWR 17x17 bundle. The bundle included 93 non instrumented and 68 instrumented heated rods. The axial power had a chopped cosine shape. The dimensions of the bundle represent a full scale PWR assembly. There are spacer grids which are located 50.0 cm (1.64 ft) apart along the axial axis of the rods. The diameter of the heater rods is 0.95 cm (0.374 in.) FLECHT SEASET measurements for the droplet size and distribution show that the droplet size distributions can be approximated by the upper-limit log normal function. Some of the reflood correlations currently used in COBRA-TF are derived using FLECHT reflood data. These correlations are described in Chapter 3.

Rod Bundle Heat Transfer (RBHT) [23] reflood tests were performed and are being performed by PennState University for Nuclear Regulatory Commission. The bundle used for these tests has 45 heater rods and four unheated corner rods. The heater rods have a 3.66 m (12.0 ft) heated length with a skewed axial power profile. Peak power is located at 2.74 m (9.0 ft) elevation. There are 7 spacer grids in the bundle and 14 thermocouples measure grid temperature at different locations.

The experiments have been performed at pressures of 137.9, 275.8 and 413.7 kPa (20, 40 and 60 psia) for the RBHT. The inlet flooding rate ranges from 0.0254 m/sec to 0.15 m/sec (1.0 in/sec to 6.0 in/sec). For high inlet reflooding cases the initial peak temperature of the heater rods is 1200.0 K (1700.0 °F) and for the low inlet reflooding cases it is 1040.0 K (1400 °F). The coolant inlet subcooling has a range between 11.1 K to 83.4 K (20 °F to 150 °F). The data from the reflood experiments with low inlet subcooling and low inlet reflood rate showed that a large portion of the bundle was covered with DFFB. These experiments provide very useful data to observe and model the DFFB heat and mass transfer mechanisms and effects of spacer grids. In addition, the wide range of experimental conditions will enable to use the RBHT data in order to validate the models improved for different DFFB heat and mass transfer mechanisms.
Yadigaroglu et al. [24] have performed studies to compare the models developed for the DFFB. Yadigaroglu et al. suggested using more than two fields for the vapor-drop flow and they recommended taking radial change in droplet distribution and steam temperatures into account. They indicated the importance of having detailed spacer grid models, especially a model simulating the droplet break up.

Ganic and Rohsenow [6] have derived a correlation for dispersed droplet heat transfer using the results of the experiments performed with nitrogen. In the heat transfer relation developed, radiation heat transfer, single phase vapor convection and droplet wall contact heat transfer were taken into account. For the radiation heat transfer, the model by Sun et al [4] was applied, and for the convection heat transfer between the rods and the vapor the Dittus-Boelter [25] correlation was used. In order to derive a model for the drop-wall contact heat transfer, the trajectories of droplets in thermal boundary layer, drop deposition flux and drop mass cumulative deposition factor were determined.

Yao and Sun [10] modeled DFFB in two regions: an entry region where droplet-wall contact occurs and a region of homogenous non-slip droplet flow. For the second region, by using the data from low reflooding tests of University of California Berkeley reflood experiments, they observed that the dispersed flow heat transfer under low reflooding conditions depend on the vapor velocity. Yao and Sun used an augmented heat transfer only for the entry region.

Using the data from FLECHT-SEASET experiments, Lee, Wong and Hochreiter [26] modeled heat transfer in DFFB. A network for radiation heat transfer was obtained and the analysis showed that, depending on the time and elevation, the radiation heat transfer might account for 20 to 75 percent of the total heat flux. Also, it was found that the droplets play an important role in enhancing the heat transfer due to increased interfacial shear between the vapor and dispersed drop phases.

Kianjah [27] observed the increase in heat transfer due to increased interfacial shear for the dispersed flow with solid particles. Kianjah performed experiments with air
and spherical glass particles in 4-rod bundle with and without flow blockage. Spherical glass diameters were 30 µm and 100 µm. The results of the analysis for dispersed flow showed that, with the 30 µm particles in the flow, the heat transfer coefficients increased and the increase was as much as 200%. The increase in heat transfer with 100 µm particles was 30%. When the Reynolds number for air flow was increased, the effect of solid particles in increasing the heat transfer was observed to diminish. When the slip ratio increased, heat transfer increased. The increase in heat transfer was observed downstream of the blockage in this study too.

Bajorek and Young [7] have used the Thermal Hydraulic Test Facility data in order to present the importance of modeling drop-wall contact heat transfer accurately for DFFB. By considering the effect of turbulence and the boundary layer thickness they modified the Forslund and Rohsenow [28] drop-wall contact heat transfer coefficient.

Ganic and Mastanaiah [29] investigated the droplet deposition from a turbulent gas stream. They developed a model for the deposition of large particles by a free flight towards the wall after diffusing from the turbulent core. The theory they developed showed that the dimensionless deposition velocity depends on Reynolds number and dimensionless relaxation time. The model developed tested with the experimental data and the agreement with the data and the model was satisfactory.

As mentioned earlier, one of the most important characteristics of DFFB is the thermal non-equilibrium between the phases. Therefore, models developed for heat and mass transfer between phases are important for DFFB modeling. Unal et al. [5] formulated a vapor generation source function for dispersed flow. The data model comparison was performed for the modified two-region model using the data from single tube and rod bundle experiments. As the result, improvements were achieved in predicting the trend of the vapor superheat data.

Yadigaroglu, Andreani [2] investigated the possibility of improving the models used in best estimate codes for reflooding. The study includes the review of the literature
and constitutive relations used reflood modeling the reflood and especially the DFFB. In conclusion, they recommended a detailed investigation of droplet field in DFFB.

### 2.2 Experimental Studies and Models Developed for the Effects of the Spacer Grids

The effects of spacer grids on heat and mass transfer during DFFB are introduced in Chapter 1. These effects were observed and measured as the results of several experiments. This section includes the summary of a survey for the data and observations obtained during the experiments and the models developed.

Marek and Rehme [30] performed rod bundle tests with spacer grids and with smooth and roughened surfaces. The change in heat flux and temperatures were measured for single phase flow. The study is one of the first studies showing the effects of spacer grids in increasing the heat transfer. Marek and Rehme developed a new correlation for heat transfer coefficient which depends on Reynolds number, Nusselt number and the flow blockage ratio.

Flood Experiments with Blocked Arrays (FEBA) [15] experiments were performed in Germany. In these experiments the effects of grids on heat transfer, heater rod temperatures and vapor temperatures were measured. Experiments were performed with a spacer grid at the mid-plane and the same experiments were repeated without a grid. The data for the experiments with and without a grid showed that, the heat transfer downstream of the grid is higher due to the convective enhancement, droplet break up and grid rewetting. The vapor temperatures measured downstream of the grid are reduced. Data showed that the heat transfer enhancement due to flow blockage decreases as the distance from the spacer grid increases [31].

The results of FEBA tests showed that the presence of the spacer grid improve cooling more when the quench front is far from the grid. The data also showed a big dip in vapor temperature downstream of a quenched grid. This shows the effect of increased interfacial heat transfer between the quenched, colder grid and the vapor.
A series of experiments were performed to investigate the effects of spacer grids on post critical heat flux flow regimes by Cluss [32]. The experiments performed with a tube having spacer grid and constant wall temperature. In the experiments, inlet flooding rate was 0.0254 m/sec (1.0 in/sec). The initial wall temperatures ranged from 810.7 K to 977.4 K (1000 to 1300 °F). The theoretical model focused on the increase in vapor velocity due to area change because of spacer grids and increase in heat transferred by radiation from wall to liquid deposited on colder spacer grid. The experimental data showed that the heat transfer is three times larger downstream of the grid.

The effects of spacer grids on heat transfer in DFFB were also observed during Oak Ridge Thermal Hydraulic Test Facility (THTF) tests [33]. The data showed a temperature depression of 75 to 150 K (135 to 270 °F) from the upstream to the downstream of the grid. This decrease in temperature corresponds to 20 % increase in heat flux. The data also showed that the spacer grids change the heat transfer mode from film boiling to transition boiling and complete rewet.

The Japan Atomic Energy Research Institute conducted series of reflood experiments with a large range of high pressure and inlet mass flux with a 5x5 rod bundle. The effects of spacer grids on quench front propagation were investigated [34]. The data obtained show that the spacer grids significantly affected the quench front propagation under high mass flux and high pressure conditions. Because of the conditions for reflood in these cases, the DFFB was observed for only very short time.

Kanazawa et al. [35] modeled the turbulence intensity and the coolant droplet behavior downstream of the spacers by using the turbulent flow analysis code. Turbulence was calculated with the $k$-$e$ model, and droplet trajectories were obtained with a Lagrangian method in consideration of the drag force from gas flow. The droplet deposition characteristics onto the fuel rod were evaluated as the coefficient by the ratio of the droplet deposition rate to the droplet concentration. The experiments performed by Kanazawa et. al. with air-water also showed an increase in gas (air) turbulence.
downstream of the grid. Data showed a maximum value just downstream from the spacer edge, and velocity decreased with increasing distance from the edge.

Several experiments were conducted to investigate the effect of a turbulent swirl generated downstream of the grid. The experiments performed by Kreith and Sonju [14] showed that, the swirl decays to about 10-20% of its initial density in a distance of about 50 pipe diameters. The decay is more rapid at smaller Reynolds number cases. Yoder [36] stated that approximately 30 hydraulic diameters are required for turbulent swirl to dissipate in single phase flow.

Shen et. al. [16] observed the effect of mixing vanes on flow in a subchannel. They observed a flow vortex created by mixing vane. Koszela observed an increase in local heat transfer downstream of the grid when a spacer grid with mixing vanes were used for low flooding rate-high power experiments [17]. Kanazawa et al. [35] showed that mixing vanes caused a different droplet deposition rate on the rods in the same subchannel downstream of the grid because of their three dimensional shapes.

A Study of Droplet Hydrodynamics across the Grid Spacer Geometry [37] was performed by State University of New York at Stony Brook for Nuclear Regulatory Commission. In this study, for air-water system, the droplet sizes and velocities upstream and downstream of 3 spacer grids were measured. In a plexiglass flow channel, water droplets were supplied by the equally spaced nozzles at the bottom of the channels. There were two types of spacer grids used in the study; one standard KWU design spacer grid and the spacer grids used in FLECHT-SEASET study. The flow channel was located between a 15mW, He-Ne laser and photo detectors. In order to measure droplet sizes and velocities, a Laser-Doppler anemometer was used.

As the result of the study, the Sauter mean diameters of large and small droplets downstream of the grid were formulated. In addition, fractional volume of small droplets downstream of the grid was calculated. The results showed that, when air velocity was low, droplets could spend more time on grids. In this case, with the same incoming
droplet velocity the small droplets breaking from the trailing edge created by film entrainment mechanism were smaller.

Several experiments with heated grid strips were performed by Carnegie Mellon University for Westinghouse to characterize the droplet break up due to impingement on a hot surface [19]. Two strips with different thicknesses were used. The strips were heated with a torch beyond the Leidenfrost temperature, then the droplets ejected from a needle from the top to the heated surface. High speed movies were made and several photographs were taken to visualize the droplet splashing. The small droplet diameters and volume ratio of large and small droplets after splashing were reported. The relation used in COBRA-TF for impinging and splashing droplet diameters was evaluated as the result of this study. The Sauter mean diameters for small and large droplets after splashing were also calculated.

Ireland et. al. [38] performed droplet size and velocity measurements upstream and downstream of a grid during RBHT experiments. The data obtained showed a good agreement with the correlation developed for the droplet break up using FLECHT-SEASET data. The data also showed that the relationship between the droplet velocity and size downstream of the grid was not significant.

Pedersen [39] and Karl et. al. [40] have also performed experiments to investigate droplet break up mechanisms when droplet contacts with a heated surface. Pedersen showed that even the small droplets break up upon impingement at moderate approach velocities. Pedersen’s analysis indicates that approach velocity is the dominant variable affecting droplet heat transfer.

Paddock [21] performed a photographic investigation of droplet behavior in dispersed flow passing through a spacer grid. The change in size and distribution of droplets entering and exiting a simulated nuclear reactor grid and the effect of the entrance length on the two phase flow pattern were photographically examined. The experiments performed for water and the quality range was 40-70%. The results of this
study showed that the entrance effect was small. The photographs showed the swirl downstream of the grid. The increase in number of drops downstream of the grid was also observed using the photographs indicating the drop break up due to spacer grids.

Stosic [41] has developed a DFFB model for a code called HECHAN. In this model, he took augmentation in heat transfer due to spacer grid blockage into account. In addition droplet break up was modeled for the drops having a size larger than the critical one. The heat transfer from the heated wall to vapor, from superheated vapor to the liquid and from wall to drop due to contact were also modeled.

Another study for droplet deposition was performed by Yano et al. [42]. Yano et al. examined the effect of spacer geometry on the droplet deposition rates downstream of the spacer in an open rectangular channel. The results of the air-water experiments and analysis showed that the deposition rate of droplets increased as the spacer grid clearance became wider and as the spacer became thicker.

The models of heat and mass transfer for DFFB and the effects of spacer grids used in COBRA-TF best estimate code were developed using the data from FLECHT-SEASET experiments. Yao, Hochreiter, Leech [43] and Chiou, Hochreiter, Utton, Young [44] proposed models representing the different heat transfer mechanisms for augmented heat transfer for different kinds of spacer grids. The model was developed for both single phase and post CHF dispersed flow. The model developed is described in Chapter 3.

Kelly and Hochreiter [12] developed the grid spacer heat transfer models used in COBRA-TF. These models are described in Chapter 3.
2.3 Studies for Droplet Hydrodynamics

The studies performed for the droplet hydrodynamics include the studies performed for droplet break up, droplet size and droplet distribution. The entrained droplets in two phase flow may breakup due to spacer grids, flow blockages or they may break up during their flight due to the forces acting on them. Ardron and Hall [45] performed reflood tests in a tube and observed droplet break up, up to 1.0 meter above quench front. Since large droplet break up is the most important source for the small droplets, the droplet break up during flight must be taken into account for DFFB modeling.

The droplet break-up studies were mainly performed for the combustion analysis and sprays. Droplet hydrodynamics in DFFB during bottom reflooding was investigated and analyzed by several researchers for flow in tubes.

In series of experiments performed for dispersed flow, two types of break up mechanisms were observed:

Aerodynamic (bag type) break up: In this case, the droplet deforms to a disk shaped droplet, and then the center of the droplet opens like a bag. Later, when the bag becomes several times larger than the original drop, a shower of droplets leaves the large drop at the bag side and the other parts of the original drop breaks into several droplets.

Capillary (stripping type) break up: This mechanism was observed infrequently during reflood. This break up occurs when the Weber number is higher, and when the liquid filaments split into large spherical droplets and a shower of small drops torn from the surface of the drop. Stripping type break up is observed when the shear on the drop surface is at a rate much faster than the rate at which drop inertia lets the drop deform as a single drop.

Weber number is the most significant parameter to model the droplet break up due to the relative motion of the droplet in a gas stream. The Weber number is a measure
of the relative importance of the inertial force compared to the surface tension force and is given as:

\[ We = \frac{\rho_l (U_v - U_d)^2}{\sigma} D_{drop} \]  

(2-1)

Many studies have been performed to determine the critical Weber number for a break up to occur. Many researchers showed that the critical Weber number is between 3.0 and 12.0 for aerodynamic break up to occur. The experiments performed Ferrenberg et. al. [46] with a gradually increasing gas velocity indicated that the critical Weber number is 10 for the low viscous flow. Later, Hinze [47] performed experiments for low viscosity fluids and found a critical Weber number of approximately 13.0. Ardron and Hall suggested a critical Weber number between 10 and 20 for aerodynamic drop break up to occur. DeSalve and Panella set the critical Weber number as 2.0 [13].

Sarjeant [48] explained the scatter in experimental results for the critical Weber number by discussing the effects of other physical phenomena like the difference in the densities of continuum phase and dispersed phase. The effects of the viscous and disruptive acceleration forces were also discussed. The analyses by Sarjeant and other researchers showed that the critical Weber number is the best criterion for the droplet break up, however, it is not the sufficient requirement. Before drops break up, drop deformation occurs and during the deformation, a break up time is elapsed. Therefore, drop has to be exposed to the gas flow for a sufficient time before it breaks up.

For the different types of break up mechanisms, different correlations for break up time have been derived. The break up time is a function of the Weber number and according to several researchers the general form of the break up time is:

\[ t_b = K \left( \frac{D_{drop}}{U_v - U_d} \right) \left( \frac{\rho_l}{\rho_g} \right)^{0.5} \]  

(2-2)
The K in above equation is a constant for which several researchers develop different values for. For the different types of the break up mechanisms, K is different and therefore it depends on the Weber number.

Kocamustafaogullari, Chen, Ishii [49] have developed a simple correlation to predict the maximum stable droplet size in a fluid. Using Kelvin-Helmholtz instability theory, which allows a relative motion at the interface, a model was developed to simulate the break up of drops and bubbles falling or rising through a fluid. The results of the calculations performed with the developed model were compared with the data from several tests performed by several researchers. The agreement between the data and calculations is favorable. Trabold, Kumar and Vassallo [50] have conducted a series of experiments with R-134a in a vertical duct. The relation derived by Kocamustafaogullari et al. for the maximum stable droplet size and Sauter mean diameter was tested and was proven to be accurate.

Ardron and Hall [45] approximated droplet size distributions by the upper-limit log normal function. As mentioned earlier, data from FLECHT-SEASET experiments showed the same distribution for the droplet size.

Smith and Azzopardi [51] investigated and summarized the results of several studies performed for the droplet sizes and size distributions. Although there are many studies and available results to use, Smith and Azzopardi suggested more experiments for variable flow conditions and flow systems in order to obtain reliable predictions.

2.4 Comprehensive Models for DFFB

These studies are based on derivation of continuity equations for two phase flow for specific geometries and solving these equations using proper constitutive relationships for heat and mass transfer mechanisms related with DFFB. The effects of spacer grids are taken into account in some of the below mentioned comprehensive studies.
Chawla and Ishii [52] derived a two-fluid model of two-phase flow for a rod bundle. The area averaged equations were derived for the flow. In this study, transverse velocities of the phases were assumed small when compared with the axial velocity. They also assumed that the change in axial velocity within a channel was stronger than the variation along the radial direction. In conclusion, they derived six equations for the flow.

For one dimensional flow, Wallis et. al. [53] derived the continuity equations and solved the equations for an evaporating mist. Thermodynamic equilibrium was assumed for the two phases. Wallis et. al. suggested using different energy equations for the liquid and vapor phases to model the rate of phase change accurately.

Sugawara and Miyamoto [54] performed calculations to compare the predictions of FIDAS code with experimental data. FIDAS code models the two phase flow as three fluid-three fields and uses Eulerian solution scheme. The code solves three continuity, three energy and six momentum equations for conservation of mass, energy and axial and lateral momentum. The constitutive relations used in the code are described in [54]. The effects of spacer grids are not modeled in FIDAS.

Three of the widely used best estimate codes, TRACE, (TRAC/Relap Advanced Computational Engine), TRAC (Transient Reactor Analysis Code, is almost entirely replaced by TRACE) and RELAP5/MOD3 codes model the two phase flow as two fields, i.e. the droplet field is not modeled as a separate field. The effects of spacer grids on heat transfer are modeled in TRAC and when a spacer grid exists the vapor to liquid heat transfer coefficient is modified for the film boiling.

COBRA-TF has three fields to model vapor, liquid film and drop phases in addition to the non-condensable gas field. The code has the ability of modeling all the effects of spacer grids observed during FEBA, FLECHT-SEASET and RBHT experiments. COBRA-TF also solves an interfacial area transport equation for the drop field. The models and equations used in COBRA-TF are described in Chapter 3 in detail.
All above mentioned best estimate codes use Eulerian solution scheme for all fields.

Andreani [55] developed a detailed Eulerian vapor field-Lagrangian droplet field, three-dimensional model for DFFB. He developed the model for tube geometry. The radial distribution of the drops in the tube and the drop break up during flight were modeled. For the drop size distribution upper limit-log normal distribution was adopted. Since the model is developed for tube geometry, spacer grid effects on the flow were not considered.

Eulerian vapor-Lagrangian drop solution scheme for rod bundles was used by Nagler [56]. The drops were grouped into up to 121 groups. The results of the model were compared with the data from reflood experiment FLECHT-SEASET test 31504. The effects of spacer grids on shattering the droplets were taken into account. However, the droplet break-up during flight was not modeled. The results show good agreement with experimental data.

Williams [57] added two droplet fields to the Eulerian-Eulerian solution scheme of the TRAC code. The mass and momentum equations for these additional fields were added to the solution and energy equation for the liquid phase was modified since the drop fields and the liquid film assumed to be at the same temperature. Two interfacial area transport equations were solved for the drop fields. The droplet break up during flight and due to spacer grids was not modeled. In addition, the change in large drop size due to evaporation was not defined. Therefore the mass and interfacial area transfer from large drop field to small droplet field was not taken into account.
2.5 Summary of the Literature Survey and Purpose of the Present Study

The literature survey performed for the DFFB and effects of spacer grids show the need for the below improvements in modeling DFFB:

1) A comprehensive model should not neglect any of the phenomenon related with the heat and mass transfer in DFFB.

2) In simulating DFFB during reflood, the models that were derived under reflood conditions must be used.

3) The empirical models that are derived using one set of experimental data must be tested using a separate set of data. The models which show good agreement with both sets of data must be used in DFFB simulation.

4) The models accounting for the detailed description of drop hydrodynamics must be included in DFFB model.

5) All the aspects of spacer grid effects on the heat and mass transfer must be modeled. Especially the entrainment and deposition of drops due to liquid film formed on spacer grids must be taken into account.

6) The comprehensive models must have proper numerical scheme to solve DFFB problem accurately and efficiently.

7) In order to model the interfacial heat and mass transfer between the continuous (vapor) and dispersed (drop) phases and to model the effect of spacer grids accurately, the drop phase must be modeled by more than one field; the number of drop fields which can be separated by their sizes must be limited due to computational constraints and effective and accurate numerical scheme must be chosen to simulate these fields.

The purpose of the present thesis study is to fill the need for above mentioned improvements for the DFFB simulation with a comprehensive approach. The current
study aims to modify the four-field solution scheme of COBRA-TF code by adding a small droplet field using the suitable models for the constitutive relationships.
Chapter 3

CURRENT APPROACH TO MODEL DFFB AND SPACER GRID EFFECTS IN COBRA-TF

The COBRA-TF (COolant Boiling in Rod Arrays-Two Fluid) computer program was developed to provide best-estimate thermal hydraulic analyses of a light water reactor vessel for design basis accidents and anticipated transients. In cooperation with FLECHT SEASET and RBHT programs, COBRA-TF was modified to enhance its predictive capability for reflood transients.

As mentioned earlier, since COBRA-TF already has an entrained drop field, it is the most suitable best estimate code to add the small droplet field. In this chapter, the COBRA-TF solution scheme, i.e. the separate time averaged transient conservation equations and closure relationships used to solve these equations are described. The models developed for the closure relationships derived for the heat transfer are also summarized. The closure relationships used for the mass and momentum transfer for dispersed droplet film boiling are described in Chapter 4 with the new models proposed.

The derivation of the average conservation equations are presented first in this section. The multiphase flow modeling used to derive the COBRA-TF equations are listed briefly.

3.1 Multiphase Flow Modeling

Multiphase flows are modeled by all best estimate codes using time averaged conservation equations for the phases. Multiphase flows can be any combination of fluids separated by the moving interfaces. Because of their complicated behavior, multiphase flows are very hard to model and it is not possible to solve exact conservation equations.
Since the best estimate codes do not aim to investigate the exact motion or behavior of one phase, the average conservation equations are derived and used in these codes.

These average equations can be derived by using separate conservation equations for each phase. The model that uses separate conservation equations is called two fluid model and in order to solve the average equations, additional models are necessary. COBRA-TF uses the equations derived by using the two fluid model and the suitable closure relationships to solve the equations. The conservation equations used in COBRA-TF and the other best estimate codes are time averaged. The Eulerian time averaging is used to calculate the variables continuous in time for a given computational node.

The derivation of the two fluid equations used in COBRA-TF has been given by Thurgood et al. [58] and is summarized below.

### 3.1.1 Exact Integral Equations for Two Phase Flow

Exact integral equations for the conservation of mass, momentum and energy are derived by using Leibnitz’s integral theorem for two phases [58].

Figure 3-1 presents the two phase volume of interest. In the figure, I(t) presents the interfacial surface between the phases and V(t) is the volume.

The mass conservation equation is:

$$\frac{d}{dt} \int_{V(t)} \rho d \bar{x} = 0$$  \hspace{1cm} (3-1)

where $\bar{x}$ represents the location.

Using Leibnitz’s rule and Gauss’ theorem, the mass conservation equation becomes,
\[
\int \frac{\partial \rho}{\partial t} \frac{dx}{v(t)} + \int \rho (u(t) dV + \int (\rho_{1} - \rho_{2})(n_{1} U_{1}) ds = 0
\] (3-2)

where \( U \) is the fluid velocity and \( \rho \) is the fluid density and \( S \) is the portion of \( I(t) \) in volume \( V(t) \).

Figure 3-1: The Two Phase Volume used in Derivations.

The conservation of momentum equation has all the forces acting on the flow:

\[
\frac{d}{dt} \int_{v(t)} \rho U dx = \int_{v(t)} \rho f dx + \int_{\partial v(t)} \tau ds + \int_{\partial v(t) I(t)} \Sigma dl
\] (3-3)

Here, \( f \) is body force per unit mass, \( \tau \) is stress vector and \( \Sigma \) is the surface tension vector. After applying the Leibnitz’s theorem, the momentum equation becomes:
Finally the equation for the conservation of energy can be written as below:

\[
\frac{d}{dt} \int_{V(t)} \rho (e \frac{1}{2} |U|^2) d\mathbf{x} = \int_{\partial V(t)} -\mathbf{Q} \cdot d\mathbf{s} + \int_{\partial V(t)} \dot{Q} d\mathbf{x} + \int_{\partial V(t)} \rho \mathbf{f} \cdot \mathbf{U} d\mathbf{x} + \int_{\partial V(t)} \nabla \cdot \mathbf{U} d\mathbf{l}
\]

where \( e \) is the specific internal energy and \( \mathbf{Q} \) is the conduction heat flux vector and \( \dot{Q} \) is the volumetric heat generation rate.

Once the Leibnitz’s theorem is applied, the energy equation becomes:

\[
\frac{d}{dt} \int_{V(t)} \rho (e \frac{1}{2} |U|^2) d\mathbf{x} + \int_{\partial V(t)} \nabla \cdot (\rho \mathbf{U}) d\mathbf{x} + \int_{\partial V(t)} \mathbf{Q} \cdot d\mathbf{s} + \int_{\partial V(t)} \dot{Q} d\mathbf{x} + \int_{\partial V(t)} \rho \mathbf{f} \cdot \mathbf{U} d\mathbf{x} + \int_{\partial V(t)} \nabla \cdot \mathbf{U} d\mathbf{l}
\]

The exact integral equations are not used to solve two phase flow as they presented above. Instead, they are used to derive conditions which represent the interaction between the phases at the interface. These conditions are called the interfacial jump conditions. Interfacial jump conditions are necessary to derive the closure relationships related to the interfacial interaction terms like interfacial mass, momentum and heat transfer.

In order to evaluate the two phase flow equations, the single phase equations are derived first then, these equations are averaged to obtain the equations for the two fluid
model. As the results of averaging the single phase equations, the jump conditions become necessary to represent the two fluid behavior.

3.1.2 Differential Single Phase Equations

The differential equations for the single phase flow is derived by applying the divergence theorem to Equations (3-2), (3-4) and (3-6) [58].

Once the divergence theorem is applied to Equation (3-2), the mass Equation becomes:

\[
\int_{V(t)} \left[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) \right] d\mathbf{x} = 0 \tag{3-7}
\]

In an arbitrary volume like \( V(t) \) and with a continuous integrand, in order to satisfy Equation (3-7) the \( \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) \) term must be zero:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \tag{3-8}
\]

Equation (3-8) is valid for either phase.

In order to derive the differential momentum equation for either phase, the stress vector in Equation (3-4) is re-written in terms of the stress tensor:

\[
\mathbf{t} = n T
\]

Here, the stress tensor can be represented by the pressure and viscous parts:

\[
T = -P I + \tau \tag{3-10}
\]
The second term in Equation (3-10) is the viscous stress tensor. Once Equation (3-10) is inserted in to Equation (3-4) and the divergence theorem is applied, the momentum equation becomes:

$$\frac{\partial}{\partial t} (\rho U) + \nabla \cdot (\rho U U) = \rho f - \nabla P + \nabla \tau$$  \hspace{1cm} (3-11)

The energy equation is derived by inserting Equations (3-9) and (3-10) and subtracting the dot product of the momentum equation and phase velocity from the resulting equation. Once these are done, the energy equation becomes:

$$\frac{\partial}{\partial t} (\rho e) + \nabla \cdot (\rho e U) = -\nabla Q + \dot{Q} - P \nabla U + \tau : \nabla U$$  \hspace{1cm} (3-12)

In the best estimate codes, enthalpy is used in energy equation:

$$h = e + \frac{P}{\rho}$$  \hspace{1cm} (3-13)

The energy equation becomes:

$$\frac{\partial}{\partial t} (\rho h) + \nabla \cdot (\rho h U) = -\nabla \cdot (Q + \mathbf{q}^T) + \dot{Q} + \mathbf{U} \cdot \nabla P + \frac{\partial P}{\partial t} + \tau : \nabla U$$  \hspace{1cm} (3-14)

As mentioned earlier, in order to solve the mass, momentum and energy equations, additional equations are necessary. The thermodynamic equations of state, the conduction vector and the stress tensor must be defined by constitutive equations.

The thermodynamic equations of state for the density and the enthalpy are written in terms of the pressure and the temperature:

$$\rho = \rho(P,T)$$  \hspace{1cm} (3-15)

$$h = h(P,T)$$  \hspace{1cm} (3-16)
In order to calculate the conduction, Fourier’s law for heat conduction is used:

\[ Q = -k(T) \Delta T \]  

(3-17)

where \( k \) represents the thermal conductivity and \( T \) is the temperature.

The stress deviator representing the shear stress in Equations (3-12) and (3-14) is defined for a Newtonian fluid as below:

\[ \tau = \lambda (\nabla U)^T + \mu (\nabla U + (\nabla U)^T) \]  

(3-18)

where \((\nabla U)^T\) represents the turbulence.

In Equation (3-18), \( \mu \) is viscosity and \( \lambda \) is second viscosity coefficient.

With the constitutive equations summarized above, the single phase flow can be simulated. For two-phase flow simulations, equations derived for the single phase are used with averaging and with the jump conditions. The jump conditions usually come about by writing a mixture equation which then force interfacial terms to cancel.

The jump conditions for the mass, momentum and energy equations are derived by manipulating Equations (3-2), (3-4) and (3-6) by using a small material volume bounding the interface for the two phases. The interfacial jump condition for the mass equation is:

\[ \rho_1 (\vec{U}_1 - \vec{U}_1) \cdot \vec{n}_1 = \rho_2 (\vec{U}_2 - \vec{U}_1) \cdot \vec{n}_1 \]  

(3-19)

The above equation for the conservation of mass at the interface shows that the rate at which mass leaves one phase is equal to the rate at which mass enters the other phase.
The derivation of jump conditions for the momentum and energy equations involves setting a jump condition for the stress tensor and defining the internal energy in terms of enthalpy. The details of this derivation can be found in [58].

By ignoring the surface tension and the viscous contribution to the pressure, the jump condition for the momentum equation becomes:

\[
P_{2z} - P_{1z} = \dot{m}_{1z} \left( \frac{1}{\rho_{1z}} - \frac{1}{\rho_{2z}} \right)
\]

(3-20)

The thermal energy jump condition is:

\[
\dot{m}_{1z} \left[ h_{1z} + \frac{1}{2} \left| U_{1z} - U_{2z} \right|^2 - \frac{1}{\rho_{1z}} (\tau_{1z} \cdot n_{1z}) n_{1z} \right] - \left[ h_{2z} + \frac{1}{2} \left| U_{2z} - U_{1z} \right|^2 - \frac{1}{\rho_{2z}} (\tau_{2z} \cdot n_{2z}) n_{2z} \right] - (Q_{1z} - Q_{2z}) n_{1z} = 0
\]

(3-21)

### 3.1.3 Time Average Equations

As mentioned earlier, the best estimate codes aim to solve the average of motion of a phase in a multiphase flow situation. In order to do that these codes use time averaged variables and equations.

In order to obtain averages for the multiphase flow variables, below relation is used for a piecewise continuous function having a piecewise derivative:

\[
\tilde{f}(x, t) = \frac{1}{\Delta t} \int_{t-\Delta t/2}^{t+\Delta t/2} f(x, t') dt'
\]

(3-22)

For the multiphase flow a phase function is defined to isolate the contribution to the average from each phase:
\[ \beta(x, t) = \begin{cases} 1 & \text{if at phase } k \text{ at time } t \\ 0 & \text{otherwise} \end{cases} \] (3-23)

The averages for the variables which can be represented with piecewise continuous functions are derived by using above phase function and the results of averaging these variables are summarized in Table 3-1.

The variables that are defined only on the phase interface are averaged by using a different methodology. In this methodology, a surface area concentration due to the \( j \)th interface and the total surface area concentration becomes:

\[ \frac{1}{L_s} = \sum_j \frac{1}{L_j} \] (3-24)

Here, the sum is over all interfaces that pass given location during a given time interval \( \Delta t \). The average for a surface variable \( \psi \) is then defined as:

\[ \overline{\psi_{i}}, = L_s \left( \sum_j \frac{\psi_{i,j}}{L_j} \right) \] (3-25)

The COBRA-TF interface variables averaged using Equation (3-25), are summarized in Table 3-2.

In addition to the average variables in Tables 3-1 and 3-2, the turbulent fluxes should also be defined. Using the methodology given by Thurgood [58], the \( k \)-phase turbulent stress tensor is

\[ \alpha_k T_{k} = -\beta_k \rho \overline{U'U'} \] (3-26)

and the \( k \)-phase turbulent heat flux is

\[ \alpha_k q_{k} = -\beta_k \rho \overline{h'U'} \] (3-27)
As mentioned earlier, the equations for the phases are derived by averaging the single-phase equations introduced. While the time averaging is performed, the expressions relating the average of a derivative to the derivative of an average variable are needed. In order to obtain such expressions following relations are used:

\[
\frac{\partial}{\partial t} (\bar{\beta}_k \psi) = \frac{\partial}{\partial t} (\bar{\beta}_k \psi) - \frac{1}{\Delta t} \sum_j \frac{1}{\nu_j} \bar{\psi}_{k,j} (\bar{\nu}_k U_j)
\]  

(3-28)

\[
\bar{\nabla} (\bar{\beta}_k \psi) = \bar{\nabla} (\bar{\beta}_k \psi) + \frac{1}{\Delta t} \sum_j \frac{1}{\nu_j} [\bar{\nu}_j \psi_{k,j} (\bar{x}, t_j)]
\]  

(3-29)

where \( \nu_j \) is the normal displacement speed of the \( j \)th interface to pass location \( \bar{x} \) during the period, \( \Delta t \).

<table>
<thead>
<tr>
<th>Definition</th>
<th>Time Averaged Variable</th>
</tr>
</thead>
<tbody>
<tr>
<td>Average k-phase void fraction</td>
<td>( \bar{\alpha}_k = \bar{\beta}_k )</td>
</tr>
<tr>
<td>Average k-phase density</td>
<td>( \bar{\alpha}_k \rho_k = \bar{\beta}_k \rho_k )</td>
</tr>
<tr>
<td>Average k-phase velocity</td>
<td>( \bar{\alpha}_k U_k = \bar{\beta}_k \bar{U} )</td>
</tr>
<tr>
<td>Average k-phase stress tensor</td>
<td>( \bar{\alpha}_k T_k = \bar{\beta}_k \bar{T} )</td>
</tr>
<tr>
<td>Average k-phase stress deviator</td>
<td>( \bar{\alpha}_k \tau_k = \bar{\beta}_k \bar{\tau} )</td>
</tr>
<tr>
<td>Average k-phase pressure</td>
<td>( \bar{\alpha}_k P_k = \bar{\beta}_k \bar{P} )</td>
</tr>
<tr>
<td>Average k-phase enthalpy</td>
<td>( \bar{\alpha}_k h_k = \bar{\beta}_k \bar{h} )</td>
</tr>
<tr>
<td>Average k-phase heat flux vector</td>
<td>( \bar{\alpha}_k Q_k = \bar{\beta}_k \bar{Q} )</td>
</tr>
</tbody>
</table>
Using Equations (3-28) and (3-29) the derivatives of the void fraction can be derived:

\[ \nabla \alpha_k = -\frac{1}{\Delta t} \sum_j \frac{1}{v_j} n_k \]

\[ \frac{\partial \alpha_k}{\partial t} = \frac{1}{\Delta t} \sum_j \frac{1}{v_j} n_j U_{ij} \] (3-30)

<table>
<thead>
<tr>
<th>Definition</th>
<th>Average Variable</th>
</tr>
</thead>
<tbody>
<tr>
<td>k-phase mass supply per unit interface area</td>
<td>( m_k = \bar{m}_{k,i} )</td>
</tr>
<tr>
<td>k-phase mass supply per unit volume</td>
<td>( \Gamma_k = \frac{m_k}{L_s} )</td>
</tr>
<tr>
<td>the average interface velocity</td>
<td>( \bar{m}<em>k U^i = \bar{m}</em>{k,i} \bar{U}_{ki} )</td>
</tr>
<tr>
<td>k-phase limit velocity</td>
<td>( \bar{m}<em>k U^i = \bar{m}</em>{k,i} \bar{U}_{ki} )</td>
</tr>
<tr>
<td>k-phase limit stress tensor</td>
<td>( T^i = T_{\equiv k, \equiv k} )</td>
</tr>
<tr>
<td>k-phase pressure</td>
<td>( P_k = \bar{P}_{ki} )</td>
</tr>
<tr>
<td>k-phase stress deviator</td>
<td>( \tau^i = \bar{\tau}_{ki} )</td>
</tr>
<tr>
<td>interfacial heat input to phase k (per unit volume)</td>
<td>( q_{ik} = -\frac{1}{L_s} (\bar{n}<em>k \bar{Q}</em>{ki}) )</td>
</tr>
<tr>
<td>k-phase limit enthalpy</td>
<td>( \bar{m}<em>k h^i_k = \bar{m}</em>{k,i} h_{ki} )</td>
</tr>
</tbody>
</table>
Using above defined averaging rules to average Equation (3-8) and using the definition of $\Gamma_k$ from Table 3-2, the average k-phase conservation of mass equation is derived as:

$$\frac{\partial}{\partial t}(\alpha_k \rho_k U_k) + \nabla.(\alpha_k \rho_k U_k) = \Gamma_k \tag{3-32}$$

In words Equation (3-32) can be explained as:

Rate of change of mass of phase $k$ + Rate of mass efflux of phase $k$ = Rate of mass transfer to phase $k$ from the other phases

Averaging Equation (3-11) to obtain the averaged k-phase momentum equation gives:

$$\frac{\partial}{\partial t}(\alpha_k \rho_k U_k) + \nabla.(\alpha_k \rho_k U_k) = \alpha_k \rho_k f_k - \nabla(\alpha_k \rho_k) + \nabla[\alpha_k (\tau + T^T_k)] - M_k \tag{3-33}$$

The average k-phase interfacial exchange of momentum can be written in a compact form as below:

$$M_k = M^I_k + P^I_k \nabla \alpha_k + M^d_k \tag{3-34}$$

Here, $M^I_k$ represents the momentum supplied to phase $k$ due to change of phase, and $P^I_k \nabla \alpha_k$ is the force on the phase $k$ due to the average interfacial pressure. The term $M^d_k$ is the drag force given as:

$$M^d_k = \frac{1}{\Delta t} \sum_j \frac{1}{\nu_j} \left[ \frac{1}{\nu_j} \right] - \mathbf{n}_k \cdot \mathbf{e}_{k_j} \tag{3-35}$$
When Equation (3-34) is inserted into Equation (3-33), the resulting equation contains a $(P_k' - P_k)\nabla\alpha_k$ term, which represents the force due to the difference between the average pressure in the bulk of phase $k$ and that at the interface. Lahey et al., improved models for virtual mass and lift forces included in $(P_k' - P_k)\nabla\alpha_k$ [59]. However, since the sum of virtual mass and lift forces is dissipative, COBRA-TF assumes that this difference is zero. In addition, COBRA-TF applications show that the only body force on the flow is the gravity. By taking these assumptions into account, the average phasic momentum equation becomes

$$\frac{d}{dt} (\alpha_k \rho_k U_k) + \nabla.(\alpha_k \rho_k U_k U_k) = \alpha_k \rho_k g - \alpha_k \nabla P + \nabla.[\alpha_k (\tau + \mathbf{T}^T)] + \mathbf{M}^T + \mathbf{M}^d$$

(3-36)

In words:

- Rate of change of momentum of phase $k$ + Rate of efflux of momentum of phase $k$
- = Gravity force + Pressure gradient force + Viscous and turbulent forces
- + Momentum exchange due to mass transfer to phase $k$ + Interface drag force

In order to obtain the average energy equation, Equation (3-14) is used with several assumptions. First of all, in COBRA-TF, the flows modeled do not generate heat. In addition, the dissipation and spatial derivative of the pressure are neglected. Therefore the Equation (3-14) can be re-written as:

$$\frac{d}{dt} (\rho h) + \nabla.(\rho h U) = -\nabla Q + \frac{\partial P}{\partial t}$$

(3-37)

Averaging above equation gives:

$$\frac{d}{dt} (\alpha_k \rho_k h_k) + \nabla.(\alpha_k \rho_k h_k U_k) = -\nabla.[\alpha_k (Q_k + q_k^T)] + \Lambda_k + \alpha_k \frac{\partial P}{\partial t}$$

(3-38)
where

\[
\Lambda_k = -\frac{1}{\Delta t} \sum_j \frac{1}{\nu_j} (\rho_{k_j} h_{k_j}) j (U_{k_j} - U_j) j n_k - \frac{1}{\Delta t} \sum_j \frac{1}{\nu_j} (n_k Q_{k_j}) j
\]  

(3-39)

representing the average k-phase interfacial supply of energy. Equation (3-39) can be re-written using Tables 3-1 and 3-2:

\[
\Lambda_k = \Gamma_k h_k^i + q_{k_i}
\]  

(3-40)

Once above equation is inserted into Equation (3-38) the resulting average phasic equation for energy becomes

\[
\frac{\partial}{\partial t} (\alpha_k \rho_k h_k) + \nabla \cdot (\alpha_k \rho_k h_k U_k) = -\nabla \cdot [\alpha_k (Q_k + q_k^T)] + \Gamma_k h_k^i + q_{k_i} + \alpha_k \frac{\partial P}{\partial t}
\]  

(3-41)

The terms in the energy equation are defined as:

Rate of change of enthalpy + Rate of efflux of enthalpy =
Conduction and turbulent heat flux
+ Energy exchange due to mass transfer to phase k
+ Interfacial heat transfer + Pressure work

3.2 COBRA-TF Two Fluid, Three Field Equations

As mentioned earlier, the two-fluid formulation uses a separate set of conservation equations (mass, energy and momentum) for each phase. COBRA-TF extends this treatment to three fields: vapor, continuous liquid and entrained liquid drops. Defining the liquid phase as two fields (drops and liquid film) enables program to handle modeling liquid film and droplet behavior reasonably in cocurrent and counter current flows. In film, inverted annular and dispersed droplet flows, the motions of droplet field and continuous liquid can be very different. Because of this difference, using one set of
average equations for liquid phase will not be good enough to model vapor and liquid interaction and liquid flow.

In addition to these three fields, COBRA-TF has the capability of modeling the transport of non condensable gas mixture with the vapor field as the fourth field as the solution scheme.

In COBRA-TF four field formulation, there are four continuity equations, three momentum equations and two energy equations. For the energy equations a thermal equilibrium is assumed between the drop and liquid film fields within a computational cell, i.e., the drop field does not carry a separate equation. This approximation is not good enough to model condensation.

The continuity equations are derived by using the equations (3-32), (3-36) and (3-41) with simple assumptions.

3.2.1 Interfacial Exchange Terms

The mass, momentum and energy exchange terms between the cases are defined using different notations for each term. In order to derive the field equations these terms and notations used must be defined. The terms describing the mass transfer between the phases are:

\[ \Gamma'' = \text{average rate of vapor generation or condensation per unit volume} \]

Since both liquid fields can contribute to the vapor generation rate, then

\[ \eta = \text{the fraction of the total vapor generation coming from the entrained liquid} \]

\[ \Gamma_e'' = \text{average rate of vapor generation or condensation per unit volume coming from the entrained liquid}, \quad \Gamma_e'' = \eta \Gamma'' \]
\[ \Gamma_i'' = \text{average rate of vapor generation or condensation per unit volume coming from the continuous liquid, } \Gamma_i'' = (1-\eta)\Gamma'' . \]

The two liquid fields can exchange mass by entrainment as well as by phase change.

\[ S'' = \text{average net rate of entrainment per unit volume} \]

With the above definitions the mass transfer terms can be written as

\[ \begin{align*}
\Gamma_v &= \Gamma'' \\
\Gamma_i &= -\Gamma_i'' - S'' = -(1-\eta)\Gamma'' - S'' \\
\Gamma_e &= -\Gamma_e'' + S'' = -\eta\Gamma'' + S'' 
\end{align*} \quad (3-42) \]

In Equation (3-42), v, l and e refer to the vapor, continuous liquid and entrained drop fields, respectively.

The interfacial momentum exchange terms are:

\[ \begin{align*}
M_i^d &= -\tau_{i,li} - \tau_{i,ei} \\
M_i^l &= -\tau_{i,il} \\
M_i^e &= -\tau_{i,ei} 
\end{align*} \quad (3-43) \]

where

\[ \tau_{i,li} = \text{average drag force per unit volume by the vapor on the continuous liquid} \ (v_i) \]

\[ \tau_{i,ei} = \text{average drag force per unit volume by the vapor on the entrained liquid} \ (v_e) \]

The momentum exchange due to mass transfer between the three fields can be written as:
\[ M^T_v = (\Gamma^+ U) \]
\[ M^T_i = -(\Gamma^+ U) - (S^- U) \]
\[ M^T_e = -(\Gamma^+ U) + (S^- U) \]

(3-44)

where \( U \) represents the velocity and detailed definition for \( U \) is given in Equations (3-62) through (3-66) in Section 3.3.1.

### 3.2.2 COBRA-TF Assumptions for Three Field Modeling

To obtain the three-field model, the following assumptions are needed:

1. The turbulent stresses and turbulent heat flux of the entrained phase can be neglected:

\[
T^{T}_{ee} = 0, q^{T}_{ee} = 0
\]

(3-45)

2. When the equations are solved on a finite-difference grid, the viscous stresses can be partitioned into wall shear and fluid-fluid shear. The fluid-fluid shear can be neglected in the entrained liquid phase. For the continuous liquid and vapor phases fluid-fluid shear is taken into account by the turbulence model. The relations used become:

\[
\nabla.(\alpha_v \tau_{ee}) = \tau^v_{ee} = 0.0
\]
\[
\nabla.(\alpha_v \tau_{vl}) = \tau^v_{vl} + \nabla.(\alpha_v \alpha^v_{vl})
\]
\[
\nabla.(\alpha_i \tau_{il}) = \tau^l_{il} + \nabla.(\alpha_i \alpha^l_{il})
\]

(3-46)

In Equation (3-46), \( \tau^v_{vl} \), \( \tau^v_{il} \) and \( \tau^l_{il} \) are the forces exerted by the wall on the vapor, the entrained liquid and the continuous liquid, respectively; \( \alpha^v_{vl} \) and \( \alpha^l_{il} \) are the fluid-fluid viscous stress tensors for the vapor and the liquid.
3. The conduction heat flux can be partitioned into a wall term and a fluid-fluid conduction term. The latter is assumed negligible in the entrained liquid. Therefore,

\[- \nabla \cdot (\alpha_v Q_v) = -\nabla \cdot (\alpha_v q_v^T) + Q_{wv}^-
\]

\[- \nabla \cdot (\alpha_c Q_c + \alpha_l Q_l) = -\nabla \cdot (\alpha_l q_l^T) + Q_{wl}^-\]

(3-47)

where \(Q_{wv}^-\) and \(Q_{wl}^-\) are the wall heat transfer rates per unit volume to the vapor and liquid, respectively; \(q_l^T\) is the turbulent heat flux for the continuous liquid; and \(q_v^T\) is the turbulent heat flux for the vapor.

4. All mass entering or leaving a phase interface is at saturation. Therefore,

\[h_v^i = h_g\]

\[h_l^i = h_f\]

(3-48)

3.2.3 Conservation Equations

Mass conservation equations are derived for vapor, continuous liquid, droplet fields and non condensable gas. These equations are:

\[\frac{\partial}{\partial t} \alpha_v \rho_v + \nabla \cdot (\alpha_v \rho_v \vec{U}_v) = \Gamma_v'' + \nabla \vec{G}_v^T\]

(3-49)

\[\frac{\partial}{\partial t} \alpha_c \rho_c + \nabla \cdot (\alpha_c \rho_c \vec{U}_c) = -\Gamma_c'' - \nabla \vec{S}_c'' + \nabla \vec{G}_c^T\]

(3-50)

\[\frac{\partial}{\partial t} \alpha_l \rho_l + \nabla \cdot (\alpha_l \rho_l \vec{U}_l) = -\Gamma_c'' + \nabla \vec{S}_c''\]

(3-51)
\[
\frac{\partial}{\partial t} \alpha_g \rho_g + \nabla \cdot (\alpha_g \rho_g \vec{U}_v) = \Gamma_g'' + \nabla \cdot \vec{G}_g^T 
\]  \hspace{1cm} (3-52)

In words, the individual terms can be explained as:

Rate of change of mass + Rate of mass gain by convection =
Rate of mass gain by interfacial transfer or chemical reaction
+ Rate of mass gain by entrainment + Rate of mass efflux due to void drift

The energy conservation equation for continuous liquid and entrained drop phases is given in Equation (3-53). The vapor- non condensable gas mixture energy equation is given in Equation (3-54).

\[
\frac{\partial}{\partial t} \alpha_v \rho_v h_v + \nabla \cdot (\alpha_v \rho_v h_v \vec{U}_v) = \Gamma'' \cdot H_g + q_{iv} + Q_{wv}'' - \nabla \cdot (\alpha_v q_v^T) 
\]  \hspace{1cm} + \alpha_v \frac{\partial P}{\partial t} 
\]  \hspace{1cm} (3-53)

\[
\frac{\partial}{\partial t} (\alpha_i + \alpha_e) \rho_i h_i + \nabla \cdot (\alpha_i \rho_i h_i \vec{U}_i) + \nabla \cdot (\alpha_e \rho_e h_e \vec{U}_e) = \Gamma'' \cdot H_f + q_{it} 
\]  \hspace{1cm} + Q_{lt}'' - \nabla \cdot (\alpha_l q_l^T) + (\alpha_i + \alpha_e) \frac{\partial P}{\partial t} 
\]  \hspace{1cm} (3-54)

In words, the terms are:

Time rate of change + Convection = Energy transport due to phase change
+ Interfacial heat transfer + Wall heat flux - Turbulent heat flux
+ Pressure derivative

The turbulent heat flux is defined in Equation (3-27).

Momentum equations are derived for three fields allowing the droplet and continuous liquid fields flow with different velocities. The equations are given as follows:
\[
\frac{\partial}{\partial t} \alpha_v \rho_v \vec{U}_v + \nabla \cdot (\alpha_v \rho_v \vec{U}_v \vec{U}_v) = -\alpha_v \nabla P + \alpha_v \rho_v \vec{g} + \vec{\tau}_{vw}''
\]
\[-\vec{\tau}_{lv}'' - \vec{\tau}_{ev}''' + (\Gamma'' \vec{U}_v)\] (3-55)

\[
\frac{\partial}{\partial t} \alpha_i \rho_i \vec{U}_i + \nabla \cdot (\alpha_i \rho_i \vec{U}_i \vec{U}_i) = -\alpha_i \nabla P + \alpha_i \rho_i \vec{g} + \vec{\tau}_{wi}'''
\[+ \vec{\tau}_{li}'' - (\Gamma'' \vec{U}_i) - (S''' \vec{U}_e)\] (3-56)

\[
\frac{\partial}{\partial t} \alpha_e \rho_e \vec{U}_e + \nabla \cdot (\alpha_e \rho_e \vec{U}_e \vec{U}_e) = -\alpha_e \nabla P + \alpha_e \rho_e \vec{g} + \vec{\tau}_{we}'''
\[+ \vec{\tau}_{le}''' - (\Gamma''' \vec{U}_e) + (S''' \vec{U}_e)\] (3-57)

Rate of change of momentum + Rate of momentum change by convection =
Pressure gradient + Gravity force + Wall shear + Interfacial drag between vapor and continuous liquid + Interfacial drag between vapor and drops
+ Interfacial momentum exchange
+ Momentum exchange due to entrainment

In order to solve these equations, closure physical models are necessary. These physical models are necessary for the mass and momentum exchange among the three fields, the drag forces at wall boundaries, for continuous fields the knowledge of turbulence terms and the entrainment rate.

In addition to the conservation equations, COBRA-TF solves an interfacial area equation to determine the interfacial area of the entrained droplet field

\[
\frac{dA^E_{vd}}{dt} + \nabla \cdot (A^E_{vd} \vec{U}_e) = A^E_{vd,E} + A^E_{vd,F}\] (3-58)

The interfacial area determined by this equation is used in energy equations to determine the interfacial heat transfer between the entrained droplet and vapor phases. It is also used in momentum equations of the entrained and vapor fields to model the
3.3 COBRA-TF Numerical Solution Method

In COBRA-TF, the equations are solved for computational meshes selected large enough to make solution fast and small enough to obtain a detailed simulation of the flow. The velocities are solved for the edges of each mesh and the state variables, i.e., density, pressure, enthalpy and volume fractions are obtained at the cell center. In order to obtain such a solution scheme, the momentum equations are solved on a staggered mesh where the momentum cell is centered on the mesh, which bounds the cell in which the scalar continuity and energy equations are solved. The staggered and scalar continuity mesh cells are presented in Figure 3-2 [60].

Once the meshes are defined using Cartesian coordinates, the finite difference equations are written for the computational cells.

---

**Figure 3-2**: COBRA-TF Mesh Cells [60]
3.3.1 Finite Difference Equations

The finite difference equations are written by using the old time and new time variables. The superscript \( n \) in below equations represents the quantities that are evaluated at the old time. The donor cell quantities evaluated at the old time carry the superscript \( \tilde{n} \). For the new time quantities, no superscript is used. The axial cross sectional area of the mesh cell is represented by \( A \) and the height of the cell is represented by \( \Delta X \) terms. In addition \( L \) represents the width of the connection between the cell and the adjacent mesh cells. Using these notations, the conservation equation for the vapor phase is written as [58]:

\[
\left[ (\alpha, \rho ), \left( \frac{\rho}{\alpha} \right)^n \right]_{A_j} \frac{\Delta t}{\Delta X_j} = \sum_{KB=1}^{NB} \left[ (\alpha, \rho ), \left( \frac{\rho}{\alpha} \right)^n U_{v_j, A_{m_j}} \right]_{KB} \sum_{KA=1}^{NA} \left[ (\alpha, \rho ), \left( \frac{\rho}{\alpha} \right)^n U_{v_j, A_{m_j}} \right]_{KA}
\]

\[
+ \sum_{KL=1}^{NKK} L_{KL} \left[ (\alpha, \rho ) \right]_{LA} + \frac{\Gamma_j}{\Delta X_j} + \frac{S_{cv_j}}{\Delta X_j} \]

Liquid mass equation becomes:

\[
\left[ (\alpha, \rho ), \left( \frac{\rho}{\alpha} \right)^n \right]_{A_j} \frac{\Delta t}{\Delta X_j} = \sum_{KB=1}^{NB} \left[ (\alpha, \rho ), \left( \frac{\rho}{\alpha} \right)^n U_{v_j, A_{m_j}} \right]_{KB} \sum_{KA=1}^{NA} \left[ (\alpha, \rho ), \left( \frac{\rho}{\alpha} \right)^n U_{v_j, A_{m_j}} \right]_{KA}
\]

\[
+ \sum_{KL=1}^{NKK} L_{KL} \left[ (\alpha, \rho ) \right]_{LA} \left( 1-\eta \right) \frac{\Gamma_j}{\Delta X_j} - \frac{S_{j}^n}{\Delta X_j} + \frac{S_{cj}}{\Delta X_j} \]

Entrained drop mass equation is:

\[
\left[ (\alpha, \rho ), \left( \frac{\rho}{\alpha} \right)^n \right]_{A_j} \frac{\Delta t}{\Delta X_j} = \sum_{KB=1}^{NB} \left[ (\alpha, \rho ), \left( \frac{\rho}{\alpha} \right)^n U_{v_j, A_{m_j}} \right]_{KB} \sum_{KA=1}^{NA} \left[ (\alpha, \rho ), \left( \frac{\rho}{\alpha} \right)^n U_{v_j, A_{m_j}} \right]_{KA}
\]

\[
+ \sum_{KL=1}^{NKK} L_{KL} \left[ (\alpha, \rho ) \right]_{LA} \eta \frac{\Gamma_j}{\Delta X_j} + \frac{S_{j}^n}{\Delta X_j} + \frac{S_{cj}}{\Delta X_j} \]

(3-59)
In above equations, $U$ represents the vertical velocity and $W$ represents the transverse velocity. The term $S$ is the entrainment rate and $S_c$ is the phase source term. $\Gamma$ represents the vapor generation. Subscript $j$ identifies the vertical location of the cell. Similarly, subscript $k$ represents the location of the cell in the horizontal plane. The total number of transverse connections to the cell is $N_{KK}$ and the total number of connections to the top of the cell is $NA$ and to the bottom of the cell is $NB$. On the right hand side of the equation the only variable evaluated at the old time is convected quantity $(\alpha, \rho, \gamma)$. The velocity evaluated at the new time value is used for the convective terms.

Using the same notation the momentum equations for the phases can be written.

Vapor phase momentum equation for the vertical direction is:

$$
\frac{\left[\alpha_v \rho_v U_{v,j} - (\alpha_v \rho_v U_{v,j})^n\right]A_{m_j}}{\Delta t} \sum_{k=1}^{NB} \left[\left(\alpha_v \rho_v U_{v,k}\right)^n U_{v,k}^n\right] A_{m_k} \Delta X_j
\frac{\sum_{K=1}^{N_{KLJ}} \left[\alpha_v \rho_v U_{v,j} \right] W_{v,j}^n L_{KLB}}{2} 
+ \sum_{K_{LA}}^{N_{LA}} \left[\left(\alpha_v \rho_v U_{v,j} \right) W_{v,j}^n \right] L_{LA} \frac{\left(P_{j+1} - P_j\right)}{\Delta X_j} \alpha_{v,j}^n A_{m_j}

- K_{n,\gamma} \left(2U_{v,j} - U_{v,j}^n\right) - K_{n,J} \left[2\left(U_{v,j} - U_{v,j}\right) - (U_{v,j} - U_{v,j})^n\right]

- K_{n,\nu} \left[2\left(U_{v,j} - U_{v,j}\right) - (U_{v,j} - U_{v,j})^n\right]

- \left[\Gamma_c U_v^m - (1-\eta)\Gamma_E U_v^m - \eta \Gamma_E U_v^m\right] \frac{S_{mv_j}}{\Delta X_j} + \tau_{v,j}

(3-62)
Liquid phase momentum equation is:

\[
\frac{\left(\frac{\partial}{\partial t} + \mathbf{U}_i \cdot \nabla\right) \mathbf{U}_i}{\Delta t} = \sum_{K=1}^{N_K} \left[ \left(\frac{\partial}{\partial t} + \mathbf{U}_i \cdot \nabla\right) \mathbf{U}_i \right] A_{m_j} + \sum_{K=1}^{N_K} \left[ \mathbf{W}^n_{i,j} \right] A_{m_k} - \frac{L_{KLB}}{2} \mathbf{U}_i \cdot \nabla \mathbf{U}_i - \mathbf{g} A_{m_j} - \frac{(P_j - P_i)}{\Delta X_j} \alpha_{m_j} (\mathbf{U}_i - \mathbf{U}_j)
\]

(3-63)

The entrained droplet field momentum equation is:

\[
\frac{\left(\frac{\partial}{\partial t} + \mathbf{U}_e \cdot \nabla\right) \mathbf{U}_e}{\Delta t} = \sum_{K=1}^{N_K} \left[ \left(\frac{\partial}{\partial t} + \mathbf{U}_e \cdot \nabla\right) \mathbf{U}_e \right] A_{m_j} + \sum_{K=1}^{N_K} \left[ \mathbf{W}^n_{e,j} \right] A_{m_k} - \frac{L_{KLB}}{2} \mathbf{U}_e \cdot \nabla \mathbf{U}_e - \mathbf{g} A_{m_j} - \frac{(P_j - P_i)}{\Delta X_j} \alpha_{m_j} (\mathbf{U}_e - \mathbf{U}_j)
\]

(3-64)
In above equations, \( K_w \) and \( K_i \) terms represent the wall and interface shear coefficients, respectively. The term \( \tau^T \) represents the fluid-fluid shear. In addition, \( S_d \) is the representation of drop deposition and \( S_e \) is the representation of drop entrainment.

For the transverse direction, equations similar to the Equations (3-62) through (3-64) are written. For the transverse momentum equations, subchannel formulation can also be used and in this type of formulation, all transverse flows are assumed to occur through gaps between channels. When the subchannel formulation is used, one transverse momentum equation is solved for all the gaps for one component. It is widely used for complex, irregular geometries and is often preferred over the Cartesian geometry. When the subchannel formulation is used, the convection terms for the transverse coordinate and orthogonal gaps are neglected. In addition to these terms, fluid-fluid shear stresses are also neglected in subchannel formulation. The details on the subchannel formulation can be found in [58].

The conservation of energy equation for the vapor phase is:

\[
\left[ \alpha_v \rho_v H_v \right]_j - \left[ \alpha_v \rho_v H_v \right]_{j-1} = \frac{\sum_{K=1}^N \left( \alpha_v \rho_v H_v \right)_{j-1} U_{\nu_v} A_{\eta_{j-1}}}{\Delta X_j} - \frac{\sum_{K=1}^{NA} \left( \alpha_v \rho_v H_v \right)_{j-1} U_{\nu_v} A_{\eta_{j-1}}}{\Delta X_j} + \sum_{K=1}^{NK} L_{KL} \left[ \alpha_v \rho_v H_v \right]_{j-1} W_{\nu_v} \]

\[= \frac{\Gamma_j H_{\nu_j}}{\Delta X_j} + \frac{q_{\nu_j}}{\Delta X_j} + \frac{Q^q_{\nu_j}}{\Delta X_j} + \frac{S_{ew}}{\Delta X_j} + \frac{Q^m_{\nu_j}}{\Delta X_j} + \alpha_y (P-P^*)_j A_{\nu_j} \]  

(3-65)

For the continuous liquid and entrained drop phases, the energy equation is:
The details of evaluating the mass, energy and momentum source terms and boundary condition source terms can be found in [58]. In addition, the fluid conduction and turbulent heat flux terms are explicit. The details of their expansion are summarized by Thurgood [58], and will not be repeated here. The sum of the conduction and turbulent heat flux between two scalar cells is derived as:

\[
\frac{\left[ (\alpha_i + \alpha_e) \rho_i H_i \right] - (\alpha_i + \alpha_e) \rho_i H_i^m}{\Delta t} \left[ A_{j,1} - A_{j,2} \right] = \\
\Delta x_j \sum_{KB=1}^{NKB} \left[ (\alpha_i \rho_i H_i) \bar{U}_{j-1}^n A_{m,j-1} + (\alpha_e \rho_i H_i) \bar{U}_{c,j}^n A_{m,j} \right]_{KB} \\
\Delta x_j \sum_{KA=1}^{NA} \left[ (\alpha_i \rho_i H_i) \bar{U}_{v,j}^n A_{m,j} + (\alpha_e \rho_i H_i) \bar{U}_{c,j}^n A_{m,j} \right]_{KA} \\
+ \sum_{KL=1}^{NKL} L_{KL} \left[ (\alpha_i \rho_i H_i) \bar{W}_{k,k,j} + (\alpha_e \rho_i H_i) \bar{W}_{k,k,j} \right] \\
\frac{\Gamma_j H_{j,i}}{\Delta x_j} + \frac{q_{k,j}}{\Delta x_j} + \frac{Q'_{j,i}}{\Delta x_j} + \frac{S_{j,i}}{\Delta x_j} + \frac{Q'^{n}_{j,i}}{\Delta x_j} + \frac{\alpha_{ij} (P-P_e)}{\Delta x_j} A_{c,i} 
\]

(3-66)

where $\xi_e + \xi_e^T$ term represents the sum of thermal diffusivity and the turbulent thermal diffusivity. Similar source terms appear in the momentum equations for the subchannel j. If the liquid is in the continuous phase, mixing is calculated for both the liquid and dispersed vapor phases. If the continuous phase is vapor, turbulent effects are assumed to be negligible for the dispersed liquid as mentioned earlier. In COBRA-TF simulations, the turbulence models are used depending on the user’s choice. In general, the turbulence models are used when a severe temperature or flow gradient exists.
3.3.2 Numerical Solution

COBRA-TF solves the conservation equations using semi implicit formulation [58]. Instead of using direct matrix solver Gaussian elimination which requires long computational time for the solution, COBRA-TF solves the momentum equations first to obtain phase flow rates, i.e., phase velocities. Once the velocities are calculated, the mass and energy equations are linearized first; a Jacobian matrix is obtained as a result of this linearization. Once the system of mass and energy equations is solved, the pressure matrix is solved to obtain the new time pressure values for each computational cell.

3.3.2.1 Solution of Momentum Equations

Momentum equations are solved using the currently known variables at one time step. The semi implicit momentum equations have below form:

\[ F_l = A_1 + B_1 \Delta P + C_1 F_l + D_1 F_v \text{ (Liquid)} \]  \hspace{1cm} (3-68)

\[ F_v = A_2 + B_2 \Delta P + C_2 F_l + D_2 F_v + E_2 F_e \text{ (Vapor)} \]  \hspace{1cm} (3-69)

\[ F_e = A_3 + B_3 \Delta P + D_3 F_v + E_3 F_e \text{ (Entrained Drop)} \]  \hspace{1cm} (3-70)

In above equations, \( F_v, F_l \) and \( F_e \) are the vapor, liquid and entrained drop flow rates, respectively. The definitions of the other parameters are summarized below:

\( A_1, A_2, A_3 = \) constants for the explicit terms in the momentum equations

\( B_1, B_2, B_3 = \) explicit portion of the pressure gradient source term

\( C_1, C_2 = \) factors that multiply the liquid flow rate with the wall and interfacial drag terms.
$D_1, D_2, D_3, E_2, E_3 =$ factors that multiply the vapor and entrained drop flow rate with the wall and interfacial drag terms.

Equations (3-68) through (3-70) can be represented in matrix form as below:

$$
\begin{bmatrix}
C_1 - 1 & D_1 & 0 \\
C_2 & D_2 - 1 & E_2 \\
0 & D_3 & E_3 - 1
\end{bmatrix} \begin{bmatrix}
F_i \\
F_v \\
F_e
\end{bmatrix} = \begin{bmatrix}
-A_1 - B_1 \Delta P \\
-A_2 - B_2 \Delta P \\
-A_3 - B_3 \Delta P
\end{bmatrix}
$$

(3-71)

Once the above system of equations is solved by using Gaussian elimination, the mass flow rates for the phases are evaluated in terms of the pressure gradient across the momentum cell $\Delta P$. When the flow rates are known, the tentative velocities for the phases can be calculated. These tentative velocities are then used to solve mass and energy equations.

### 3.3.2.2 Solution of Mass and Energy Equations

The mass and energy equations are non-linear equations since they include terms which include the multiplication of the unknowns. In order to solve the mass and energy equations, the right hand side of the equation is moved to the left hand side. When the solution converges, the right hand sides of the new equations, i.e., the residual errors in the equations become zero. Equation (3-72) presents the residual error of the vapor mass equation, $E_{cv}$, as an example.

$$
E_{cv} = \frac{\left[ (\alpha_v \rho_v)_j \right] - \left[ (\alpha_v \rho_v)_{j'} \right] \Delta t}{\Delta X_j} + \sum_{KB=1}^{NB} \left[ (\alpha_v \rho_v)^\theta U_{viren} A_{miren} \right]_{KB} + \sum_{KA=1}^{NA} \left[ (\alpha_v \rho_v)^\theta U_{viren} A_{miren} \right]_{KA} \frac{\Delta }{\Delta X_j} + \sum_{KL=1}^{NN} L_{KL} \left[ (\alpha, \rho_v)^\theta \right]_{KL} \left. \frac{\Gamma_j}{\Delta X_j} + \frac{S_{cv}}{\Delta X_j} \right.
$$

(3-72)
The velocities in the above equation are the tentative velocities obtained once the momentum equations are solved.

The residual errors for the mass equations of continuous liquid, entrained drop and the non-condensable gas fields are $E_{cl}$, $E_{ce}$ and $E_{cg}$. The residual errors of the energy equations are $E_{ev}$ and $E_{el}$ for the vapor and continuous liquid-entrained drop phases.

The mass and energy equations are solved by taking the variation of each of the independent variables into account using Newton-Raphson method. In order to do this, the equations are linearized with respect to the independent variables, $P$, $\alpha_v$, $\alpha_cH_v$, $(1-\alpha_v)H_l$, $\alpha_e$ and $\alpha_g$. Once the linearization is completed, the matrix system below is obtained for the equations of mass for non-condensable gas, vapor, liquid and drop fields and energy equations:

$$
\begin{bmatrix}
\frac{\partial E_{eg}}{\partial \alpha_g} & \frac{\partial E_{eg}}{\partial \alpha_v} & \frac{\partial E_{eg}}{\partial \alpha_c H_v} & \frac{\partial E_{eg}}{\partial (1-\alpha_v) H_l} & \frac{\partial E_{eg}}{\partial \alpha_e} & \frac{\partial E_{eg}}{\partial P_j} & \frac{\partial E_{eg}}{\partial P_{i=1}} & \cdots & \frac{\partial E_{eg}}{\partial P_{i=NCON}} \\
\frac{\partial E_{el}}{\partial \alpha_g} & \frac{\partial E_{el}}{\partial \alpha_v} & \frac{\partial E_{el}}{\partial \alpha_c H_v} & \frac{\partial E_{el}}{\partial (1-\alpha_v) H_l} & \frac{\partial E_{el}}{\partial \alpha_e} & \frac{\partial E_{el}}{\partial P_j} & \frac{\partial E_{el}}{\partial P_{i=1}} & \cdots & \frac{\partial E_{el}}{\partial P_{i=NCON}} \\
\frac{\partial E_{ev}}{\partial \alpha_g} & \frac{\partial E_{ev}}{\partial \alpha_v} & \frac{\partial E_{ev}}{\partial \alpha_c H_v} & \frac{\partial E_{ev}}{\partial (1-\alpha_v) H_l} & \frac{\partial E_{ev}}{\partial \alpha_e} & \frac{\partial E_{ev}}{\partial P_j} & \frac{\partial E_{ev}}{\partial P_{i=1}} & \cdots & \frac{\partial E_{ev}}{\partial P_{i=NCON}} \\
\frac{\partial E_{ce}}{\partial \alpha_g} & \frac{\partial E_{ce}}{\partial \alpha_v} & \frac{\partial E_{ce}}{\partial \alpha_c H_v} & \frac{\partial E_{ce}}{\partial (1-\alpha_v) H_l} & \frac{\partial E_{ce}}{\partial \alpha_e} & \frac{\partial E_{ce}}{\partial P_j} & \frac{\partial E_{ce}}{\partial P_{i=1}} & \cdots & \frac{\partial E_{ce}}{\partial P_{i=NCON}} \\
\frac{\partial E_{cg}}{\partial \alpha_g} & \frac{\partial E_{cg}}{\partial \alpha_v} & \frac{\partial E_{cg}}{\partial \alpha_c H_v} & \frac{\partial E_{cg}}{\partial (1-\alpha_v) H_l} & \frac{\partial E_{cg}}{\partial \alpha_e} & \frac{\partial E_{cg}}{\partial P_j} & \frac{\partial E_{cg}}{\partial P_{i=1}} & \cdots & \frac{\partial E_{cg}}{\partial P_{i=NCON}} 
\end{bmatrix}
\begin{bmatrix}
d\alpha_g \\
d\alpha_v \\
d(\alpha_cH_v) \\
dP_i \\
\vdots \\
\end{bmatrix} = \begin{bmatrix}
E_{eg} \\
E_{el} \\
E_{ev} \\
E_{ce} \\
E_{cg} \\
\end{bmatrix}
$$

(3-73)

The matrix, containing the ratios of variation in residual errors and variation in independent variables, is called a Jacobian matrix. The variation of velocity with respect to pressure is evaluated by solving the momentum equations. The derivatives of other dependent variables are either obtained from the thermal equations of state or from fundamental identities involving partial derivatives.
Once the derivatives for the six equations are evaluated, Equation (3-73) is reduced using Gaussian forward elimination. The set of equations are reduced such that the equations involve only the pressures. This new set of equations is solved by using Gauss-Seidel iterative technique. An iterative technique is used since using a direct solution method is very impractical for a large mesh. Recently, Cuervo et al. [61] implemented SuperLU direct solution method to reduce the duration of the solution process i.e., CPU time significantly.

The convergence criterion for pressure is given as:

\[
\frac{|p^{n+1} - p^n|}{p^n} < EPSO
\]  

(3-74)

where \(EPSO\) is a user defined limit for the maximum pressure change rate in a single iteration. This convergence criterion is important particularly at low pressure.

Once the pressures are obtained, the remaining unknowns are calculated. Finally, the velocities are determined by using the new pressure gradients.

If the convergence criteria are satisfied, the iteration is completed and the code moves on and performs the calculations for the next time step. If the convergence limits are not satisfied for specified number of outer iterations, i.e., the iterations performed to solve momentum equations, then outer iteration is considered to have failed. All fluid conditions are reset to the previous time step value, the time step size is reduced by half and the calculation is repeated.

### 3.4 COBRA-TF Heat Transfer Package

The heat transfer package in COBRA-TF consists of a library of correlations and a selection logic which allows the code to predict a boiling curve as a function of the computational cell void fraction, pressure, mass flow and the heated surface.
temperature. Figure 3-3 shows the boiling curve and regions of interest and Figure 3-4 shows the heat transfer regime selection logic. The heat transfer package which is used calculates both the wall to-fluid heat transfer as well as the interfacial heat transfer between the phases. Since separate energy equations are used for the phases, a non-equilibrium flow will be calculated in some cases. Therefore, the interfacial heat transfer and the interfacial heat transfer area are calculated to determine the temperature of each phase. Both will be discussed below with the emphasis on reflood heat transfer.

1. Single Phase Vapor

COBRA-TF calculates the local Reynolds number within the computational cell and determines if the flow is laminar or turbulent. If the flow is turbulent, it uses the maximum of the Dittus-Boelter correlation [37] or the correlation developed from the FLECHT-SEASET 161-rod bundle tests [60].

If the flow is calculated to be laminar, the code uses a Nusselt number of 10, which is also based on the FLECHT-SEASET data.

2. Single Phase Liquid

In a similar fashion, the code calculates the Reynolds number of the cell and if the flow is turbulent, the code uses the Dittus-Boelter correlation for the convective heat transfer. If the flow is calculated to be laminar, the heat transfer correlation by Sparrow et al [62] is used which has a maximum Nusselt number equal to 7.86.

3. Nucleate Boiling

When the wall temperature exceeds the saturation temperature but is less than the wall temperature at the critical heat flux point, the Chen [63] correlation is used to calculate the wall heat transfer. The Chen correlation applies to both saturated nucleate boiling and forced convection evaporation and will automatically transition into single phase convection at low wall superheats and into pool boiling at low mass flow rates. The
Chen correlation regards the wall heat transfer as consisting of a combination of forced convection heat transfer as well as pool boiling heat transfer. In this fashion both extreme limits of forced convection and pool boiling are preserved.

4. Subcooled Nucleate Boiling

The Chen correlation can also be extended into the non-equilibrium regime of subcooled nucleate boiling. Again, the Chen correlation combines a forced convective heat transfer contribution and a boiling contribution to calculate the total wall heat transfer. For the subcooled case, the "F" factor used in the Chen correlation is set to unity but the remainder of the correlation is applied as in the nucleate boiling case, including the computation of suppression factor, S.

In subcooled nucleate boiling, there exists thermodynamic non-equilibrium between the voids which are formed and the bulk liquid temperature which is subcooled. Therefore, there is heat transfer between the vapor and the liquid such that the vapor condenses and the liquid temperature increases along the channel. The heat transfer processes of interest include: forced convection to the liquid, vapor generation at the wall, condensation near the wall, and bulk condensation in the liquid core.

The partition of the vapor generation and the forced convection portions of the wall heat flux are calculated by the Chen correlation for the given set of conditions. The interfacial heat transfer processes are directly calculated in the fluid energy equations as part of the fluid conditions for the cell. The, near wall condensation was calculated using the Hancox-Nicoll correlation [64] which was then subtracted from the nucleate boiling heat transfer portion to obtain the net vapor generation. There were further refinements which accounted for the fraction of the subcooled liquid which would penetrate the saturated liquid layer on the wall using the Routhani and Axelsson correlation [65]. Using this approach, the net amount of vapor generation at the wall can be calculated and the remainder of the vapor will then be mixed in to the bulk flow through the liquid energy equation and will condense.
Figure 3-3: Boiling Curve used in COBRA-TF

Figure 3-4: Heat Transfer Selection used in COBRA-TF
5. Critical Heat Flux

COBRA-TF calculates the critical heat flux and the wall temperature superheat at the CHF point to fix this location on the boiling curve as shown in Figure 3-3. For reflood heat transfer, the Zuber [66] pool boiling correlation is chosen for the critical heat flux since the liquid flow velocities are small.

6. Minimum Stable Film Boiling Point

The other point which is fixed on the boiling curve is the minimum film boiling point, $T_{\text{min}}$. This location denotes the boundary between stable film boiling and transition boiling. COBRA-TF uses the larger of a modified version of the homogeneous nucleation temperature which is curve fit as a function of the difference between the critical pressure and the local pressure, and which has also been modified to account for wall properties; and the Henry [67] correlation. In addition, for reflood, COBRA-TF limits the value of $T_{\text{min}}$ to be

$$426 \, ^\circ\text{C} (800 \, ^\circ\text{F}) < T_{\text{min}} < 650 \, ^\circ\text{C} (1200 \, ^\circ\text{F}).$$

7. Transition Boiling

The transition region is viewed as a mixture of film boiling with a vapor layer contacting the wall and nucleate boiling or wetted wall in which liquid contact with the wall is possible. The wetted wall portion of the wall heat flux is calculated using Ganic and Rohsenow [6] which uses the McCoy and Hanratty model [68] for determining the droplet migration to the wall. Once the droplet contacts the wall, a droplet efficiency is calculated which is a function of the wall temperature and the liquid temperature. At high wall temperatures, the efficiency becomes very small as the drops will not contact the hot wall. The wetted wall portion of the heat flux is added to the dry wall film boiling heat flux calculation to give the total transition boiling wall heat flux.
There are separate models for top down quench in which the heat transfer is enhanced below the top down quench front location which is used as a multiplier on the critical heat flux.

8. Inverted Annular Film Boiling

COBRA-TF assumes the wall heat transfer is in inverted annular film boiling if the wall temperature is greater than $T_{\text{min}}$, and the void fraction is less than 0.8. The modified Bromley correlation [69] is used for the film-boiling portion of the wall heat flux. The radiation heat transfer from the wall to the liquid is also accounted for as well as the droplet contact heat transfer using the Ganic and Rohsenow correlation as described earlier. Therefore:

$$q''_{\text{IAFB}} = q''_{\text{Brom}} + q''_{R} + q''_{W-D}$$  \hspace{1cm} (3-75)

When the cell void fraction is greater than 0.4 and less than 0.9, the wall heat flux is linearly interpolated between the value for inverted annular film boiling and dispersed flow film boiling. There are also interfacial heat and mass transfer models in the inverted annular film-boiling regime which include estimates of the interfacial area between the vapor and the liquid such that the proper liquid and vapor temperatures can be calculated.

9. Dispersed Flow Film Boiling

In COBRA-TF, the dispersed flow film boiling regime is selected when void fraction is greater than 0.9. It is a three-step method as described in Equation (3-76) and heat transfer to the superheated vapor is the dominant mechanism.

The total heat flux is:

$$q''_{\text{DFFB}} = q''_{FC} + q''_{R} + q''_{W-D}$$  \hspace{1cm} (3-76)

where
\( q''_{FC} \) is vapor convective heat flux

\( q''_R \) is radiation heat flux

\( q''_{w-D} \) is drop impingement heat flux.

The vapor convective heat flux is:

\[
q''_{FC} = f_{2p} f_{grad} H_{SPV} (T_w - T_f) \tag{3-77}
\]

\( H_{SPV} \) is selected as the maximum of the Dittus-Boelter correlation, FLECHT-SEASET 161-rod steam cooling correlation, and a laminar flow Nusselt number.

Dittus-Boelter correlation:

\[
HTC = 0.023 \frac{k_v}{D_H} \left( \frac{G_v D_H}{\mu_v} \right)^{0.8} Pr^{0.4} \tag{3-78}
\]

FLECHT-SEASET correlation

\[
HTC = 0.0797 \frac{k_v}{D_H} \left( \frac{G_v D_H}{\mu_v} \right)^{0.6774} Pr^{0.333} \tag{3-79}
\]

Laminar Flow:

\[
HTC = 10 \frac{k_v}{D_H} \tag{3-80}
\]

All vapor properties are evaluated at the film temperature for single phase vapor convection.

Once \( H_{SPV} \) is selected as the maximum of the above heat transfer coefficients, it is multiplied by a two-phase enhancement factor, \( f_{2p} \). Two-phase enhancement factor
represents the enhancement of the convective heat transfer due to an increase in turbulence because of the interfacial shear between dispersed particles and continuous phase.

Two-phase enhancement factor is derived by using momentum-heat transfer analogy, and it is calculated as below:

\[ f_{2b} = \left( 1 + \frac{\tau_d}{\tau_w} \right)^{\frac{1}{2}}, \tag{3-81} \]

where

\[ \frac{\tau_d}{\tau_w} = 1.5 \alpha_e \left( \frac{D_H}{D_d} C_{D_u} f_w \left( \frac{U_v - U_d}{U_v} \right) \right)^2 \tag{3-82} \]

Where, \( \tau_w \) is vapor-wall shear stress, \( \tau_d \) is interfacial shear due to droplets, \( f_w \) wall friction factor, \( C_{D_u} \) is drop drag coefficient, \( D_d \) is droplet diameter and \( \alpha_e \) is volume fraction of entrained droplets. These are the instantaneous local variables.

In the HSPV relation \( f_{\text{grid}} \) represents the heat transfer enhancement due to grids. The derivation of \( f_{\text{grid}} \) will be discussed later.

In the RBHT version of the COBRA-TF code, heat flux due to radiation to drop and radiation to vapor is added to the total heat flux. The model used for wall to drop and wall to vapor radiation heat transfer is derived by Sun et al [4], and it represents liquid, vapor and wall with an electrical network.

In the FLECHT-SEASET version of the COBRA-TF code, the drop impingement heat flux is calculated by using below relations:

\[ q''_{w-D} = S_{DE} h_{fg} \eta \tag{3-83} \]
here, \( S_{DE} \) is drop migration rate towards wall given as

\[
S_{DE} = k_d C
\]  
(3-84)

Here, \( k_d \) is drop deposition coefficient and \( C \) is the droplet concentration given by:

\[
C = \rho_v \left( \frac{G_d}{G_v} \right)
\]  
(3-85)

In the drop impingement heat flux relation, Equation (3-83), \( \eta \) represents the drop evaporation efficiency and is approximated by:

\[
\eta = \exp \left[ 1 - \left( \frac{T_w}{T_f} \right)^m \right]
\]  
(3-86)

In COBRA-TF, \( m \) is taken as 2, and in recent studies, for high-pressure situations, it is given as:

\[
\left[ \left( \frac{T_w - T_{sat}}{T_{CHF} - T_{sat}} \right)^{\frac{1}{2}} \right]^{\eta}
\]  
(3-87)

In the RBHT version of the COBRA-TF code, the correlation below is used for droplet impingement heat flux (Forslund-Rohsenow, [39]):

\[
q''_{w-D} = (0.2)(1.276)(1 - \alpha_v)^{\frac{3}{2}} \left[ \frac{g \rho_f \rho_g h_{fg}^* k_g^3}{(T_w - T_{sat}) \mu_g D_d} \right]^{\frac{1}{4}} (T_w - T_{sat})
\]  
(3-88)

The term \( h_{fg}^* \) is:
Interfacial heat transfer correlations are summarized in Equation (3-90).

\[
\frac{h^*_{fg}}{h_{fg}} = \left[ 1 + 0.35 \left( \frac{C_p (T_w - T_{sat})}{h_{fg}} \right) \right]^{3/5}
\]

\[
H_{SPV} = \frac{2 r_d}{1 + 0.5(h_v - h_f) / h_{fg}} \left( 2.0 + 0.55 \frac{Re_d^{0.5} Pr_{v}^{1/3}}{Pr} \right) k_v (\text{superheated vapor})
\]

\[
H_{SCV} = 1.0 \times 10^4 \quad (\text{subcooled vapor})
\]

\[
H_{SHL} = 2.7 \frac{\pi^2}{3} \frac{k_f}{r_d} \quad (\text{superheated liquid})
\]

\[
H_{SCL} = 2.7 \frac{\pi^2}{3} \frac{k_f}{r_d} \quad (\text{subcooled liquid})
\]

### 3.5 Small Droplet Field

In order to improve accuracy in predicting DFFB, an explicit small droplet field was added to the code. The shattered large droplets are defined as the source for small droplets. COBRA-TF uses only flow blockages and spacer grids as the cause of large drop shattering.

There are several assumptions and restrictions in COBRA-TF’s small droplet field:

1) The small drop field is not implicitly coupled with the overall hydrodynamic solution. The small droplet field interfaces with the fluid solution only through:

- as a sink/source term for vapor generation
- the removal of large droplets into the small droplets.
2) Lateral transport of small drops between channels is ignored.
3) The model is developed only for vertically upward flow.

Since the effect of a spacer grid on the droplet field is of particular interest for this thesis study, the models used in COBRA-TF for droplet break-up due to spacer grids are described here. The details of the spacer grid model are given in Section 3.6.

The grid and blockage models specify the mass source for the small drops and initial small droplet diameter. Once these parameters are known, the number flux for small droplets can be defined as:

\[ N_{SD} = \frac{S_{SD}}{\frac{\pi}{6} \rho_f D_{SD}^3} \]  \quad (3-91)

where

\[ S_{SD} = \text{mass source of small drops} \quad \text{and} \quad D_{SD} = \text{diameter of small drops} \]

The drop number flux \( N_{SD} \), number of drops per second, is assumed to remain constant downstream of the breakup point.

By using the known value for small droplet diameter, the vapor generation rate from a single drop can be found. The relation used in COBRA-TF for vapor generation rate is given in Equation (3-92) as:

\[ \Gamma = h_f \pi D_{SD}^2 \left( \frac{T_v - T_f}{h_{fg}} \right) \]  \quad (3-92)
where

\( \Gamma \) = vapor generation rate from one drop

\( h_i \) = interfacial heat transfer coefficient

\( T_v \) = superheated vapor temperature

\( T_f \) = saturation temperature

\( h_{fs} \) = latent heat of vaporization

In order to evaluate \( h_i \), the Lee-Ryley correlation is used:

\[
h_i = \frac{k_v}{D_{SD}} \left( 2 + 0.55 \text{Re}_D \text{Pr}_v \frac{Y}{X} \right)
\]  

(3-93)

where

\( k_v \) = thermal conductivity of vapor

\[
\text{Re}_D = \text{drop Reynolds number} \left( \frac{\rho_v (V_v - V_{SD}) D_{SD}}{\mu_v} \right)
\]  

(3-94)

\( \rho_v \) = superheated vapor density

\( V_v \) = axial velocity of vapor

\( V_{SD} \) = axial velocity of small drop

\( \mu_v \) = vapor viscosity
Pr_v = vapor Prandtl number

In addition, a correction for the effect of saturated vapor leaving the drop surface is applied:

\[
h_i = \frac{k_v (2 + 0.55 \text{Re}_D \text{Pr}_v)^{1/2}}{D_{SD}} \left( \frac{T_v - T_f}{1 + 0.5C_p \frac{T_v - T_f}{h_{fg}}} \right)
\]

Therefore

\[
\Gamma = \frac{k_v (2 + 0.55 \text{Re}_D \text{Pr}_v)^{1/2} \pi D_{SD}^2 (T_v - T_f)}{D_{SD} [h_{fg} + \frac{1}{2} (h_v - h_g)]}
\]

where

\[h_v = \text{vapor enthalpy}\]

\[h_g = \text{saturated vapor enthalpy}\]

This can be re-written:

\[
\Gamma = C_\Gamma D_{SD}
\]

where

\[
C_\Gamma = \frac{k_v (2 + 0.55 \text{Re}_D \text{Pr}_v)^{1/2} \pi (T_v - T_f)}{h_{fg} + \frac{1}{2} (h_v - h_g)}
\]

As above relation indicates, \(C_\Gamma\) is a function of vapor temperature and drop relative velocity.
The mass conservation equation for a drop is given as:

\[
\frac{dD_{SD}}{dZ} = \frac{2C_\Gamma}{\pi \rho_l D_{SD} V_{SD}}
\]  

(3-99)

and accounts for the drop evaporation in the axial direction.

By assuming that \( C_\Gamma \) and \( V_{SD} \) are constants in one node, the diameter of droplet leaving the node is given by

\[
D_{SD_2} = \left[ D_{SD_1}^2 - \frac{4C_\Gamma}{\pi \rho_l V_{SD}} \Delta Z \right]^{\frac{1}{2}}
\]  

(3-100)

here,

\[ D_{SD_2} = \text{exit drop diameter} \]

\[ D_{SD_1} = \text{inlet drop diameter} \]

\[ \Delta Z = \text{node length} \]

The vapor generation in a node i is:

\[
\Gamma_i = N_{SD} \frac{\pi}{6} (D_{SD_2}^3 - D_{SD_1}^3) \rho_l
\]  

(3-101)

The calculation procedure for above models can be summarized as below:

1) At a grid spacer or flow blockage, mass source, number flux and initial diameter for small drops are calculated.

2) The initial velocity is assumed to be one half of that of the impacting drop.

3) Evaporation constant, \( C_\Gamma \), is determined by using the diameter and relative drop velocity at the bottom of the node.
4) The drop diameter at the top of the node is determined from Equation (3-100)
5) The vapor generation rate is determined by using Equation (3-101)
6) The drop velocity at the top of the node is calculated by using a simplified
momentum equation:

\[ V_{SD} \frac{dV_{SD}}{dZ} = 0.75 \frac{C_D \rho_r (V_v - V_{SD})^2}{\rho_l D_{SD}} - g \]  

(3-102)

Here, all quantities are evaluated at the bottom of the node. Then,

\[ V_{SD_i} = V_{SD_0} + \left[ 0.75 \frac{C_D \rho_r (V_v - V_{SD_i})^2}{\rho_l D_{SD_i}} - g \right] \frac{\Delta Z}{V_{SD_i}} \]  

(3-103)

7) The evaporation constant for the next node is calculated by using new drop
diameter and velocity, and so forth. The new interfacial area calculated here is
coupled with the interfacial area equation, therefore is coupled with the energy
equation.
8) This process continues until all the small drops are evaporated or they exit the
bundle.

If all the small drops generated at one grid are not evaporated before the next one,
they are combined with the drops generated at the next grid. Population merging is
calculated by using conservation of mass for the entrained phase as:

\[ \left[ N_D D_{SD}^3 \right]_{i+1} = (N_D D_{SD}^3)_i + (N_D D_{SD}^3)_{i+1} \]  

(3-104)

\[ \left[ N_D D_{SD}^2 \right]_{i+1} = (N_D D_{SD}^2)_i + (N_D D_{SD}^2)_{i+1} \]  

(3-105)

Therefore

\[ [D_{SD}]_{i+1} = \frac{\left[ N_D D_{SD}^3 \right]_{i+1}}{\left[ N_D D_{SD}^2 \right]_{i+1}} \]  

(3-106)
and

\[ [N_D]_{i+1} = \frac{[N_D D_{SD}^3]_{i+1}}{D_{SD}^3}_{i+1} \] (3-107)

Drop momentum is also conserved:

\[ [V_{SD}]_{i+1} = \frac{(N_D D_{SD}^3 V_{SD})_i + (N_D D_{SD}^3 V_{SD})_{i+1}}{N_D D_{SD}^3}_{i+1} \] (3-108)

In order to couple physical models developed for small droplet field accurately with the conservation equations, instead of solving these models explicitly, and coupling the result through the interfacial area equation and therefore the energy equation, small droplet field mass and momentum equations can be added to the code’s conservation equations.

**3.6 Spacer Grid Model in COBRA-TF**

In fuel bundles, spacer grids are used in order to support the fuel rods. The effects of the grids on heat transfer can be summarized as below:

**3.6.1 Convective Enhancement**

Grids reduce the fuel assembly flow area; they cause an expansion and contraction in flow. This contraction and expansion disrupts and reestablishes thermal and fluid boundary layers. Finally, this causes an increase in the turbulence and in heat transfer from wall to continuous vapor flow.
In dispersed flow film boiling regime, the convective heat transfer enhancement affects the single-phase vapor heat transfer coefficient. In COBRA-TF code, the correlation developed by Yao, Hochreiter and Leech [43] is used. This correlation is:

\[
\frac{Nu_z}{Nu_0} = 1 + 5.55 a_r^2 \exp \left( \frac{-0.13z}{D_h} \right)
\]

(3-109)

Here, \(a_r\) is the fraction of the channel blocked by the grid. The exponential term shows the exponential decrease in heat transfer enhancement downstream of the grid.

3.6.2 Grid Rewetting

During reflood, grids can be quenched before fuel rods since they are unpowered. Therefore, liquid is deposited on the grids and can create a liquid film on them. The liquid surface on the grids cause vapor desuperheat because of increased interfacial heat transfer area.

In addition, the evaporation of the liquid film on the grid adds saturated vapor to the flow and increase the steam flow and convective heat transfer.

These two mechanisms cause a decrease in vapor temperature downstream of the grids.

In COBRA-TF, grid quenching is modeled in two regions. Below the quench front, grid is wet and this region is called wet region. In wet region, the grid spacer temperature is saturated liquid temperature. Above the quench front, grid is dry and its temperature is almost equal to rod temperature. There are different heat balance models for each region.

1) Dry region heat balance: The below heat balance is used in dry region is given as:
\[
\frac{P_g}{A_c} (q''_{rad} - q''_{conv} - q''_{dcht}) = \rho C_p \frac{dT_{grid}}{dt}
\]  

(3-110)

where

\[T_{grid} = \text{dry grid temperature}\]

\[A_c = \text{cross-sectional area}/2\]

\[P_g = \text{perimeter of grid strap}\]

\[q''_{rad} = \text{radiation heat flux from the heater rods and vapor}\]

\[q''_{conv} = \text{vapor convective heat flux}\]

\[q''_{dcht} = \text{heat flux due to drop contact}\]

For the radiation heat flux from the rods and from the vapor, geometry of the enclosure and a radiation heat flux network is used. The radiation heat flux to the dry region of the grid spacer is:

\[q''_{rad} = \frac{B_2 - \sigma T_2^4}{A_2 R_{22}} \]  

(3-111)

Here, \(B_2\) is black body radiosity of the grid spacer, \(T_2\) is grid temperature, \(A_2\) is the surface area of the grid and \(R_{22}\) is the resistance of the grid in the radiation heat flux network.

The vapor convective heat flux is calculated by using the same convective heat transfer coefficient used for the rod-vapor heat transfer:

\[q''_{conv} = h(T_{grid} - T_v). \]  

(3-112)
where $T_v$ is the vapor temperature.

2) Wet region heat balance: In the wet region, the droplets reaching the grid projection area of the grid are assumed to be captured and are added to the liquid film calculated by quench front model.

For the grid quench front calculation the difference between evaporation and liquid deposition is calculated. If liquid deposition is greater than the evaporation, the grid remains wet, otherwise, quench front will recede.

For the radiation heat transfer, the grid temperature is taken as saturation temperature and liquid emissivity is equivalent to grid emissivity. The radiation heat flux network used for dry region is also used for wet region. Figure 3-5 presents the radiation heat flux network [60].

![Radiation Heat Flux Network](image)

Figure 3-5: Radiation Heat Flux Network used for Radiant Heat Flux to the Grid [60]

Interfacial heat transfer between liquid film on the grid and vapor is calculated as below:
\[ q''_1 = (h_{\text{conv}})_g (T_v - T_f) \]  

(3-113)

where \((h_{\text{conv}})_g\) represents the convective heat transfer coefficient to the continuous vapor and is given as:

\[ (h_{\text{conv}})_g = \frac{f_I}{2 \rho_v C_p U_{vl} \Pr_v^{\frac{3}{2}}} . \]  

(3-114)

Here, \(f_I\) is the film friction factor.

### 3.6.3 Droplet Breakup Model

The grids can shatter the entrained droplets into smaller, thermally more active droplets. The small droplets easily evaporate and provide additional steam flow, therefore increase the convective heat transfer. They also desuperheat the steam.

To develop COBRA-TF FLECHT SEASET version, the results of there major experiments were used for droplet breakup model. These experiments are conducted by the The University of New York at Stony Brook [16] and Westinghouse/Carnegie Mellon University [17]. All experiments show the droplet breakup occurs when a drop impinges on a grid spacer. The shattered droplets are added to small drop field to calculate the heat and mass transfer between droplets and vapor.

In the droplet breakup model two quantities are used as inputs: the fraction of the incident drop population that is shattered into microdrops and the initial drop diameter.

The experiments performed for developing COBRA-TF showed that the break-up due to spacer grids can be defined as “slicing” of the impacting drop, and this can be illustrated as in Figure 3-6. As the result of slicing, there may be two large droplets or one large droplet with large number of small droplets. These micro droplets are added to the small
droplet field, and the increase in interfacial heat transfer due to the large droplet is ignored.

The mass of small droplets generated after the break up is defined as the function of entrained drop flow rate and the grid blockage area:

\[ m_{DB} = \eta_e \left( \frac{A_d}{A_c} \right) m_e. \]  \hspace{1cm} (3-115)

---

Figure 3-6: Droplet Slicing due to Spacer Grid
In this relation, $A_g$ is the projected grid blockage area, $A_c$ is channel area and $\eta_c$ is grid efficiency factor representing the portion of the drop that is shattered into microdroplets. FEBA and FLECHT SEASET experiments show that $\eta_c$ can be taken as 0.6. This grid efficiency is used for both large drop break up and further break up of small droplets.

The diameter of the shattered drops are evaluated by using following relations derived by using the results of above mentioned experiments,

$$We_D = \frac{\rho I_{Dy}^2 D_I}{\sigma}$$  \hspace{1cm} (3-116)

In this relation, $We_D$ is droplet Weber number, $V_{Dy}$ is perpendicular velocity of drop impacting to the surface, and $D_I$ is the diameter of the impacting drop. $D_I$ is calculated from the entrained flow and droplet number density in the upstream cell:

$$D_I = \frac{6\alpha_c}{A_{I,d}}$$  \hspace{1cm} (3-117)

From the microdrop frequency distributions obtained by the experiments, Sauter mean diameters of the microdrops were correlated as:

$$D_{SD} / D_I = 6.167 We_D^{-0.53}$$  \hspace{1cm} (3-118)

In COBRA-TF, it was assumed that when drop impact Weber number is low, the order of magnitude for the diameter of shattered drops is the same as the impacting drops and no shattering occurs. Therefore the shattered drops are large drops, and the interfacial area generated by drop break up is added to the interfacial area transport equation for drop field as an extra source term. The interfacial area transport equation is described in Section 3.7.
Figure 3-7 shows the plot of drop break up data for different drop size to grid thickness ratios as a function of Weber number of impacting drop. The curve fit to the data obtained by using FLECHT-SEASET data is also presented on Figure 3-7.

When:

\[ \text{We}_D < 150: \text{drop is added to large drop field} \]

\[ 150 < \text{We}_D < 250: \]

\[ \epsilon = \frac{\text{We} - 150}{250 - 150} \]  

(3-119)
ε \( m_{\text{SD}} \) becomes the source term for small droplet field

\( (1-\varepsilon) \ m_{\text{SD}} \) becomes source term for large droplet field

We\(_D \geq 250\): all drops are shattered and added to small drop field.

### 3.7 Interfacial Area Transport Equation

The interfacial area for the entrained droplet field that is used in energy and momentum equations is given as:

\[
\frac{dA_{i,d}^\ddagger}{dt} + \nabla \cdot (A_{i,d}^\ddagger \overrightarrow{U}) = A_i^{\ddagger E} + A_i^{\ddagger \Gamma} \tag{3-120}
\]

where Equation (3-120) relates:

- Rate of change of interfacial area concentration +
- Rate of efflux of interfacial area concentration =
- Rate of interfacial area concentration generation by entrainment and deposition
- + Rate of interfacial area concentration change due to phase change

Equation (3-120) is solved explicitly, and written as:

\[
V_c \frac{dA_{i,d}^\ddagger}{dt} = \sum_j (A_{i,d}^\ddagger \overrightarrow{U}_j A) \ j + A_i^{\ddagger E} + A_i^{\ddagger \Gamma} \tag{3-121}
\]

where the first term on the right hand side represents the net contribution of interfacial area from all axial and transverse droplet flows into the cell. The second term accounts for entrainment, deposition and droplet break up. The final term is the
contribution due to phase change and is obtained from the net evaporation rate of the droplet field.

The change in drop interfacial area due to phase change within the cell is calculated from a mass balance:

$$N_d \frac{dV_d}{dt} = -\frac{\eta \Gamma}{\rho_i}$$  \hspace{1cm} (3-122)

where \( N_d \) is the number of drops in the cell, \( V_d \) id the drop volume, and \( \eta \Gamma \) is the evaporation rate of the droplet field.

Since

$$N_d = \frac{\alpha_c V_c}{V_d}$$  \hspace{1cm} (3-123)

then Equation (3-122) can be approximated as

$$\frac{\alpha_c V_c}{V_d} (V_d - V_d^n) = -\frac{\eta \Gamma \Delta t}{\rho_i}$$  \hspace{1cm} (3-124)

Here, the old time value of the drop volume is represented with the superscript “n”, and \( V_c \) is the cell volume.

The surface area and volume can be related as,

$$\frac{V_{d, s}}{V_d^n} = \left( \frac{A_{s,s}}{A_{n,d}} \right)^{\frac{3}{2}}$$  \hspace{1cm} (3-125)

then, Equation (3-124) becomes
If break up does not occur, this equation is applied to a single droplet or \( N_d \) droplets.

The interfacial area concentration is then:

\[
A_{i,d} = A_{i,d}^{\text{m}} \left(1 - \frac{\eta \Gamma \Delta t}{\rho \alpha \epsilon V_c} \right)^{\frac{1}{2}}
\]

(3-126)

The interfacial area concentration found is then compared to possible lower limits, and the final value of \( A_{i,d} \) is selected.

The drop interfacial area for entrained flow is given by

\[
A_{i,drop} = A_{i,d}^* A_x \Delta X
\]

(3-128)

Where \( A_x \) is the flow area in the momentum cell, and \( \Delta X \) is the cell height.
Chapter 4

PROPOSED MODEL AND DISCUSSION OF THE SOLUTION

In Chapter 3, the currently used methods in COBRA-TF to simulate the heat transfer mechanisms in DFFB and effects of spacer grids are described in detail. As mentioned earlier, COBRA-TF has Eulerian-Eulerian model. The conservation equations that are currently used in COBRA-TF are shown in Equations 3-49 through 3-57.

As discussed in Chapter 3, the entrained liquid is modeled as a separate field and is divided into two droplet size groups to model the effect of smaller droplets that are thermally more active. However, the smaller droplet model is not implicitly coupled to the overall conservation equations.

The problem of handling the small droplet as a separate field arises when it comes to modeling the transformation of large droplets into small droplets. This transformation occurs due to large drop break up. The droplet break up may occur due to spacer grids and flow blockages, and it may occur while the drop is being carried by the vapor, i.e., due to capillary and aerodynamic effects. Although the break up of droplets due to spacer grids and flow blockages is modeled in COBRA-TF, the droplet break up due to capillary and aerodynamic effects are not modeled. In addition, the lateral small droplet flow is ignored.

In the existing COBRA-TF, the evaporation from small droplets is added to the vapor field mass and energy equations as a source term, but it is not included in the vapor field momentum equation; this decreases the accuracy of code’s calculations since the increase in vapor flow is not reflected in the momentum equation.
Adding the small droplet field to the flow model implicitly can improve the capability of COBRA-TF for modeling the heat and mass transfer in DFFB; therefore, is proposed in this work.

The modified equations and closure relationships that are used to solve the proposed five field model are described in this chapter. In addition, the interfacial area equations for two drop fields and modifications to the spacer grid models are summarized. Finally, necessary modifications to currently existing convective enhancement and minimum film boiling temperature models are described.

4.1 Proposed Model

In order to add the small droplet field to the COBRA-TF solution scheme implicitly, the conservation equations for this new field must be derived and implemented in the code. In addition, in order to take the mass and heat transfer between the small droplet field and the other fields into account, the conservation equations currently solved by COBRA-TF must be modified.

The model proposed in current study will have following equations:

1) Mass equation for vapor
2) Mass equation for liquid film
3) Mass equation for non-condensable gases
4) Mass equation for large droplet field
5) Mass equation for small droplet field
6) Axial and transverse momentum equations for vapor
7) Axial and transverse momentum equations for liquid film
8) Axial and transverse momentum equations for large droplet field
9) Axial and transverse momentum equations for small droplet field
10) Energy equation for vapor
11) Energy equation for liquid
Table 4-1 shows the equations to be added (presented by a + sign), and equations to be modified (presented by M).

<table>
<thead>
<tr>
<th>Equation</th>
<th>Vapor</th>
<th>Liquid</th>
<th>Large Drop</th>
<th>Small Droplet</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass</td>
<td>M</td>
<td>M</td>
<td>M</td>
<td>+</td>
</tr>
<tr>
<td>Momentum</td>
<td>M</td>
<td>M</td>
<td>M</td>
<td>+</td>
</tr>
<tr>
<td>Energy</td>
<td>M</td>
<td>M</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Interfacial Area</td>
<td></td>
<td></td>
<td>M</td>
<td>+</td>
</tr>
<tr>
<td>Transport</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

As Table 4-1 indicates, a separate interfacial area transport equation for the small droplet field will be added to the code’s solution scheme. In order to calculate the interfacial heat transfer between the small droplet field and the vapor field, the interfacial area between these two fields must be calculated. An accurate calculation of interfacial area for small droplet field also is important to simulate the interfacial drag.

Table 4-1 presents that, all the existing equations will be modified and three new equations will be introduced. The entrained drop field is divided into two fields as small droplet and large drop fields.

The non-condensable gas mass equation will remain unchanged once the small droplet field is added to the code.
4.2 Modified and Proposed Equations

As mentioned earlier, adding small droplet field model to COBRA-TF requires not only adding new equations to the conservation equations but also requires modifying other equations. The terms in conservation equations that should be changed or should be added to the solution scheme are summarized in Table 4-2. In Table 4-2, the term \( \alpha_{sm} \) represents the volume fraction for the small droplet field and \( U_{sm} \) and \( W_{sm} \) represent the small droplet vertical and transverse velocities, respectively.

The conservation equations that would be added to the solution system for the small droplet field are described in section 4.2.1 in detail.

Table 4-2 indicates that mass transfer from continuous liquid and entrained drop fields to vapor field due to evaporation must be re-modeled or current model should be modified in order to take the effect of small droplet evaporation into account. The existing COBRA-TF four field solution scheme models the evaporation of small droplets generated at the spacer grids or at the flow blockages and the small drop evaporation term is included in the vapor mass and energy equations, but it is not included in the momentum equation. In section 4.2.2 the modification in current evaporation model is described.

Besides the large drop break up at the spacer grids or flow blockages, entrainment at the quench front and entrainment from liquid film are also mass sources for small droplet field. COBRA-TF has entrainment models developed for entrainment in film flow and entrainment during reflood. The entrainment model was recently modified by Holowach [70], where two entrainment models were developed by using force balances and stability analyses for film entrainment and entrainment at the quench front. These entrainment models give the entrainment rate, but they do not define the size distribution of the entraining drops. Therefore, further analyses and assumptions are required to use this model for the five field model with two drop fields. These analyses are discussed in section 4.2.3.
Table 4-2: Modifications to Existing COBRA-TF Conservation Equations

<table>
<thead>
<tr>
<th>Equation</th>
<th>Term</th>
<th>Notation</th>
<th>Equation Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vapor Mass</td>
<td>Interfacial mass transfer due to evaporation</td>
<td>$\Gamma^*$</td>
<td>(3-49)</td>
</tr>
<tr>
<td>Vapor Energy</td>
<td>Interfacial heat transfer due to evaporation</td>
<td>$\Gamma^* h_v$</td>
<td>(3-53)</td>
</tr>
<tr>
<td>Vapor Momentum</td>
<td>Interfacial momentum transfer due to evaporation</td>
<td>$\Gamma^* \bar{U}_v$</td>
<td>(3-55)</td>
</tr>
<tr>
<td></td>
<td>Interfacial drag between vapor and small droplet field</td>
<td>$\bar{\tau}<em>{I</em>{smv}}^{**}$</td>
<td>will be added to (3-55)</td>
</tr>
<tr>
<td>Liquid Mass</td>
<td>Interfacial mass transfer due to evaporation</td>
<td>$\Gamma^*_l$</td>
<td>(3-50)</td>
</tr>
<tr>
<td>Liquid Energy</td>
<td>Rate of change of energy</td>
<td>$\frac{\partial}{\partial t} (\alpha_i + \alpha_e + \alpha_{sm}) \rho_l h_i$</td>
<td>(3-54)</td>
</tr>
<tr>
<td></td>
<td>Energy transfer due to mass efflux</td>
<td>$\nabla.(\alpha_{sm} \rho_l h_i \bar{U}_{sm})$</td>
<td>will be added to (3-54)</td>
</tr>
<tr>
<td></td>
<td>Interfacial heat transfer due to evaporation</td>
<td>$\Gamma^*_f H_f$</td>
<td>(3-54)</td>
</tr>
<tr>
<td>Liquid Momentum</td>
<td>Interfacial momentum transfer due to evaporation</td>
<td>$\Gamma^*_l \bar{U}_l$</td>
<td>(3-56)</td>
</tr>
<tr>
<td></td>
<td>Momentum transfer due to entrainment</td>
<td>$S_{sm}^{**} \bar{U}_l$</td>
<td>will be added to (3-56)</td>
</tr>
<tr>
<td>Entrained Drop Mass</td>
<td>Interfacial mass transfer due to evaporation</td>
<td>$\Gamma^*_e^{**}$</td>
<td>(3-51)</td>
</tr>
<tr>
<td>Entrained Drop Mass</td>
<td>Entrainment</td>
<td>$S^{**}$</td>
<td>(3-51)</td>
</tr>
<tr>
<td></td>
<td>Break up due to spacer grids and flow blockages</td>
<td>$S_{BDS}^{*}$</td>
<td>will be added to (3-51)</td>
</tr>
<tr>
<td></td>
<td>Break up during flight</td>
<td>$S_{BDF}^{*}$</td>
<td>will be added to (3-51)</td>
</tr>
<tr>
<td>Entrained Drop Momentum</td>
<td>Momentum transfer due to evaporation</td>
<td>$\Gamma_e^{***} \bar{U}_e$</td>
<td>(3-57)</td>
</tr>
<tr>
<td>Entrained Drop Momentum</td>
<td>Momentum transfer due to entrainment</td>
<td>$S^{***} \bar{U}_e$</td>
<td>(3-57)</td>
</tr>
<tr>
<td></td>
<td>Momentum transfer due to break up at the spacer grids and flow blockages</td>
<td>$S_{BDS}^{***} \bar{U}_e$</td>
<td>will be added to (3-57)</td>
</tr>
<tr>
<td></td>
<td>Momentum transfer due to break up during flight</td>
<td>$S_{BDF}^{***} \bar{U}_e$</td>
<td>will be added to (3-57)</td>
</tr>
</tbody>
</table>
Another mass source for the small droplet field is the large drop break up due to aerodynamic forces. As mentioned earlier, there are two mechanisms for large drop break up. Those are drop break up at due to spacer grids and flow blockages and drop break up during flight due to forces acting on the large drop. COBRA-TF models the drop break up at the spacer grids and at the flow blockages. The drop break up at the spacer grids is already described in Chapter 3. In Section 4.2.4 the large drop break up during flight is described.

The major force acting on the small droplets is the interfacial drag between the vapor and the small droplet fields. Therefore the interfacial drag between these two fields must be included in the momentum equations solved for both fields. The interfacial drag model that is used for the small droplet field is described in Section 4.2.5.

During the liquid film/drop flow regime (one of the cold-wall flow regimes in COBRA-TF), drop de-entrainment is modeled and same model is used for the small droplet de-entrainment. The de-entrainment model currently used in the code is described in Section 4.2.6. Small droplet de-entrainment is neglected for the dispersed droplet film boiling.

In Section 4.2.7 the interfacial area transport equations for large drop and small droplet fields are discussed and described.

4.2.1 Conservation Equations for Small Droplet Field

The conservation equations for the small droplet field are derived by using several assumptions. These assumptions can be summarized as below:

1) The small droplet, large drop and continuous liquid fields are assumed to be in thermal equilibrium. Therefore, there is no need to derive a separate energy equation for the small droplet field. The changes in
energy equation derived for the large drop and continuous liquid fields are presented in Table 4-2.

2) The only mass sources for the small droplet field are the entrainment and large drop break up. No additional mass source is taken into account.

3) There is no sink (or source) boundary condition for small droplet field.

4) The deposition of small droplets on the fuel rods and the structure is ignored because of the high axial velocity of small droplets. Therefore, the wall-small droplet field contact heat transfer is ignored.

5) The effect of wall shear on small droplet field is ignored

Once above assumptions are taken into account, the mass and momentum equations for the small droplet field becomes:

**Mass equation:**

\[
\frac{\partial}{\partial t} \alpha_{sm}\rho_t + \nabla \cdot (\alpha_{sm}\rho_t \vec{U}_{sm}) = -\Gamma_{sm}^- + S_{sm}^- + S_{BDS}^- + S_{BDF}^- - S_{DE}^-
\]  

(4-1)  

In words:

Rate of change of mass + Rate of mass gain by convection =  
- Rate of mass transfer due to evaporation  
+ Rate of mass gain by entrainment  
+ Rate of mass gain by large drop break up at spacergrids  
+ Rate of mass gain by large drop break up due to aerodynamic forces  
- Rate of mass loss due to de-entrainment

**Momentum equation:**

\[
\frac{\partial}{\partial t} \alpha_{sm}\rho_t \vec{U}_{sm} + \nabla \cdot (\alpha_{sm}\rho_t \vec{U}_{sm} \vec{U}_{sm}) = -\alpha_{sm} \nabla P + \alpha_{sm} \rho_t \vec{g} + \vec{\tau}_{sv}^-  
- (\Gamma_{sm}^- \vec{U}_{sm}) + (S_{sm}^- \vec{U}_t) + (S_{BDS}^- \vec{U}_e) + (S_{BDF}^- \vec{U}_e) - (S_{DE}^- \vec{U}_e)
\]  

(4-2)
Rate of change of momentum + rate of momentum change by convection =
Pressure gradient + Gravity force
+ Interfacial drag between vapor and small droplets
+ Interfacial momentum exchange due to evaporation
+ Momentum exchange due to entrainment
+ Momentum exchange due to large drop break up at spacer grids
+ Momentum exchange due to large drop break up due to forces
- Momentum exchange due to de-entrainment

The finite difference equations for the Cartesian coordinates are:

**Mass:**

\[
\begin{align*}
\frac{[\alpha_{sm}\rho_i - \alpha_{sm}\rho_j] U_{sm,j}(U_{sm,j+1} - U_{sm,j-1})}{\Delta X_j} & = \sum_{KB=1}^{NB} \left[ \alpha_{sm}\rho_i \frac{U_{sm,j} A_{m,j}}{\Delta X_j} \right]_{KB} - \sum_{KB=1}^{NA} \left[ \alpha_{sm}\rho_i \frac{U_{sm,j} A_{m,j}}{\Delta X_j} \right]_{KA} \\
& + \sum_{KL=1}^{NKK} L_{KL} \left[ \alpha_{sm}\rho_i \frac{W_{sm,j}}{\Delta X_j} \right]_{LA} - \frac{\Gamma_{sm,j} S_{sm,j}^n}{\Delta X_j} + \frac{S_{BDS}^n}{\Delta X_j} + \frac{S_{BDF}^n}{\Delta X_j} - \frac{S_{DE}^n}{\Delta X_j}
\end{align*}
\]

(4-3)

**Vertical Momentum:**

\[
\begin{align*}
\frac{[\alpha_{sm}\rho_i U_{sm,j} - \alpha_{sm}\rho_j U_{sm,j}] A_{m,j}}{\Delta X_j} & = \sum_{KB=1}^{NB} \left[ \alpha_{sm}\rho_i \frac{U_{sm,j}^n A_{m,j}}{\Delta X_j} \right]_{KB} - \sum_{KB=1}^{NA} \left[ \alpha_{sm}\rho_i \frac{U_{sm,j}^n A_{m,j}}{\Delta X_j} \right]_{KA} \\
& - \frac{\sum_{KL=1}^{NKK} L_{KL} \left[ \alpha_{sm}\rho_i U_{sm,j} \right]_{LA} W_{sm,j}^n}{\Delta X_j} - \frac{\sum_{KL=1}^{NKK} L_{KL} \left[ \alpha_{sm}\rho_i U_{sm,j} \right]_{LA}^2}{2} \frac{L_{KLB}}{\Delta X_j} \\
& + \frac{K_{i,s_m}^n \left[ \Gamma_{v} U_{sm,j} - (U_{sm,j})_{j} \right]}{\Delta X_j} + \frac{(S_{sm,j}^n U_{sm,j}^n)}{\Delta X_j} + \frac{(S_{BDS} U_{sm,j}^n)}{\Delta X_j} + \frac{(S_{BDF} U_{sm,j}^n)}{\Delta X_j} - \frac{(S_{DE} U_{sm,j}^n)}{\Delta X_j}
\end{align*}
\]

(4-4)
Transverse Momentum:

\[
\frac{\left[\alpha_{sm} \rho_j W_{sm} \right]_j - \left[\alpha_{sm} \rho_j (W_{sm})^n\right]_j L_j \Delta X_j}{\Delta \xi} = \sum_{KL=L}^{NKII} \left[\left[\alpha_{sm} \rho_j U_{sm}\right]_j W_{sm}^{in} L_{KL} \right] \Delta X_j \\
- \sum_{K=L}^{NKII} \left[\left[\alpha_{sm} \rho_j \tilde{W}_{sm} \right]_j W_{sm}^{in} L_{KL} \right] \Delta X_j \\
+ \sum_{i=1}^{NG} \left[\left[\alpha_{sm} \rho_j \tilde{W}_{sm} \right]_j W_{sm}^{in} L_{KL} \right] \Delta X_j \\
+ \sum_{i=1}^{NCB} \left[\left[\alpha_{sm} \rho_j \tilde{W}_{sm} \right]_j W_{sm}^{in} L_{KL} \right] \Delta X_j \\
- \frac{(P_{j} - P_{H})}{\Delta Z_j} \alpha_{sm} W_{sm} \Delta X_j + K_{iz, vsm} \left[2(W_v - W_{sm})_j - (W_v - W_{sm})_j + \left(\frac{\Gamma_{sm} \tilde{W}_{sm}}{\Delta X_j}\right)_j\right]
\]

In the transverse momentum equation, NKII is the number of mesh cells facing the upstream face of the mesh cell and NKJJ is the number of cells facing the downstream face of the mesh cell. The number of cells connected to the mesh cells whose velocities are orthogonal to the velocity of the cell under consideration is NG. NCA is the number of cells connected to the top of the cell, and NCB is the number of cells connected to the bottom.

In order to solve above equations, the models for mass, momentum and heat transfer should be evaluated. Following sections include the description of these models.

4.2.2 Mass Transfer due to Evaporation

In COBRA-TF calculations, vapor generation is divided into four components. The definition of these components depends on whether the liquid or vapor phase is superheated or subcooled. The total vapor generation rate is given by the sum of these components. The vapor generation per unit volume for the superheated liquid is given by:
\[ \Gamma_{\text{SHL}} = \frac{H_{\text{SHL}} A_f (h_i - h_f)}{C_{p_f} h_{fs}} \]  

(4-6)

where \( H_{\text{SHL}} \) is the interfacial heat transfer coefficient determined using proper correlations for the continuous liquid and drop fields and \( A_f \) is the interfacial area. Interfacial area for the droplet field is calculated by solving the interfacial area transport equation. For the subcooled liquid, superheated vapor and subcooled vapor, similar relations are used in COBRA-TF. The fraction of total vapor generation coming from the entrained liquid (\( \eta \)) is calculated using:

\[
\eta = \min \left[ \frac{\alpha_c}{1 - \alpha_v}, \left( 1.0 - \frac{Q_{\text{wl}}}{\Gamma h_{fs}} \right) \right] 
\]  

(4-7)

where \( Q_{\text{wl}} \) is the heat transferred from heated wall to continuous liquid field.

Once the entrained drop field is divided into large drop and small droplet fields, the model used for the vapor generation coming from the entrained liquid i.e. the Equation (4-7) must be modified.

In this study, Equation (4-7) is modified by adding the volume fraction of the small droplet field,

\[
\eta = \min \left[ \frac{\alpha_c + \alpha_{sm}}{1 - \alpha_v}, \left( 1.0 - \frac{Q_{\text{wl}}}{\Gamma h_{fs}} \right) \right] 
\]  

(4-8)

where \( \alpha_{sm} \) is the volume fraction for the small droplet field.

Equation (4-8) gives the vapor generation from both large drop and small droplet fields. In order to determine the fraction of vapor generation coming from the large drop field, below relation for the ratio of volume fractions is used:
\[ \eta_{LD} = \frac{\alpha_c}{\alpha_c + \alpha_{sm}} \eta \]  \hspace{1cm} (4-9)

Therefore, the vapor generation coming from the large drop field is:

\[ \Gamma'' = \eta_{LD} \Gamma'' \]  \hspace{1cm} (4-10)

For the small droplet field, the fraction of vapor generation is:

\[ \eta_{SM} = \frac{\alpha_{sm}}{\alpha_c + \alpha_{sm}} \eta \]  \hspace{1cm} (4-11)

Using Equation (4-11), the vapor generation from small droplet field is:

\[ \Gamma''_{sm} = \eta_{SM} \Gamma'' \]  \hspace{1cm} (4-12)

### 4.2.3 Entrainment of Small Droplets

As mentioned earlier, the entrainment model used in COBRA-TF was modified by Holowach [70]. The COBRA-TF predictions with this new model were tested with FLECHT-SEASET and RBHT experiments and code predictions showed a good agreement with the data.

Holowach’s entrainment model gives the mass rate for the entrained drops; however it does not give the size distribution for these drops. In order to calculate the mass source for large drop and small droplet fields, the entrained drop size distribution should be known.

Several researchers, for example, earlier, Mugele and Evans [71], later Ardron and Hall [45] observed that the drop size distribution can be presented using an upper limited log normal distribution for dispersed flow. The data from FLECHT-SEASET
experiments and recently, data from RBHT experiments showed an agreement with these observations.

Once the drop size distribution is known, the mean size of the drops can be evaluated in order to obtain the size of a drop which can be used to represent the entire drop field. In COBRA-TF, the Sauter mean diameter is used as the mean diameter. Sauter mean diameter is the diameter of a drop which has the same volume to surface area ratio as the entire droplet distribution. The Sauter mean diameter is given as:

\[
SMD = D_{3,2} = \frac{\int D^3 f(D) dD}{\int D^2 f(D) dD}
\]  \hspace{1cm} (4-13)

In Equation (4-13) \(NG\) is the number of droplet groups containing the similar sized droplets, \(D\) is the representative drop size for a group and finally \(f(D)\) numbers of drops per unit diameter. Integration terms in Equation (4-13) can be re-written as the summation of discrete functions:

\[
SMD = D_{3,2} = \frac{\sum_{i=1}^{NG} n_i D_i^3 \Delta D}{\sum_{i=1}^{NG} n_i D_i^2 \Delta D} = \frac{\sum_{i=1}^{NG} n_i D_i^3}{\sum_{i=1}^{NG} n_i D_i^2} = \frac{6V_d}{A_d}
\]  \hspace{1cm} (4-14)

where \(n_i\) represents the number of droplets in group \(i\), \(n_r\) is the total number of groups and \(\Delta D\) is the diameter bin width. In Equation (4-14) \(V_d\) and \(A_d\) are the total volume and total interfacial area of the drops, respectively. COBRA-TF calculates the volume fraction of the drop field once the mass equation is solved. The interfacial area for the drop field is calculated by solving the interfacial area transport equation. Therefore, the Sauter mean diameter is easily calculated by COBRA-TF using Equation (4-14).
In order to determine the volume and surface area of the large drop and small droplet fields, Williams [57] used the drop size distribution derived by Kataoka, Ishii and Mishima [71]. Kataoka et al. correlated the droplet size distribution using a volume fraction oversize, which is defined as the volume fraction of the droplets having a diameter larger than D.

\[
\frac{d\Delta}{dy} = -\frac{0.884}{\sqrt{\pi}} e^{-0.781y^2}
\]  

(4-15)

where

\[
y = \ln\left(\frac{2.13D}{D_{\text{max}} - D}\right)
\]  

(4-16)

For the mean droplet size, Kataoka et al. evaluated a volume median diameter, which represents a volume oversize of 50 percent. Kataoka et al. evaluated a correlation for the volume median diameter using data from several experiments. The drop size distribution they used and correlation for the volume median diameter showed that the ratio of the maximum drop size and volume median diameter is:

\[
\frac{D_{\text{max}}}{D_{\text{vm}}} = 3.13
\]  

(4-17)

and the ratio of Sauter mean diameter and volume median diameter is:

\[
\frac{SMD}{D_{\text{vm}}} = 0.8
\]  

(4-18)

Using Equations (4-15) through (4-17), Williams [57] evaluated and tabulated the volume fraction and area fraction of the drops having diameters ranging between \(D_o\) to \(D_{\text{vm}}\). The volume fractions were obtained by integrating (4-15) and (4-16). As the result of the integration, for different \(D/D_{\text{vm}}\) ratios, the volume fractions of the small droplets
generated by the entrainment were evaluated. The tabulated results showed that, when the diameter is the Sauter mean diameter, the volume ratio of small droplets is 34 percent.

For the COBRA-TF model, these tabulated results were used to determine the volume fraction i.e. the mass fraction of small droplets generated by the entrainment. In order to determine volume median diameter, Equation (4-18) is used. As mentioned earlier, COBRA-TF calculates the Sauter mean diameter by solving the conservation of mass and interfacial area transport equations. Once the Sauter mean diameter is known, the volume median diameter can be easily evaluated using (4-18).

The diameter $D$ is evaluated using the droplet data from RBHT experiments. Here, $D$ represents the diameter of smallest drop in the large drop field. Therefore, it is an upper limit for the diameter range for the small droplet field. For the RBHT data, this diameter limit is determined by using the Weber number criterion currently used in COBRA-TF. This criterion is given by Equation (3-119).

During RBHT experiments, the drop sizes and number of drops having the same size were measured. The drop velocities were measured, however the measurements were at the downstream of a spacer grid. Therefore, the drop velocity data from RBHT experiments can not be used to evaluate the Weber numbers which will be used in Equation (3-119).

In order to determine the relative drop velocities, a simple force balance for a single drop is used [56]. For the RBHT drop size measurements, only gravity and drag forces are taken into account. For the drag force per unit mass below relation is used:

$$\tau = 0.375 \frac{C_d}{D} \frac{\rho_v}{\rho_d} |U_d|$$

where $C_d$ is the drag coefficient given as:
\[ C_d = \frac{24}{Re} \left[ 1.0 + 0.1(\text{Re})^{0.75} \right] \]  

(4-20)

and Reynolds number is:

\[ \text{Re} = \frac{\rho_v U_d D}{\mu_v} \]  

(4-21)

In Equations (4-19) through (4-21) \( U_d \) is the relative drop velocity and \( \rho_v \) and \( \mu_v \) are the vapor density and viscosity. In Equation (4-19) \( \rho_d \) represents the density of the droplet.

Once the drag force per unit mass is calculated, a force balance can be used to determine the relative velocity of the drop:

\[ g = \tau U_d \]  

(4-22)

where \( g \) is the gravitational acceleration.

Equations (4-19) through (4-22) are solved using an iterative process. The iteration starts with an initial guess for relative drop velocity and continues until the solution of (4-22) for the relative velocity converges. Once the relative drop velocity is obtained, the Weber numbers of the drops with the diameters measured during RBHT tests are calculated.

Table 4-3 presents a sample Weber number calculation performed for the RBHT data. Table 4-3 contains the data for droplet size measurements, the relative drop velocities obtained by using above mentioned force balance and the Weber numbers calculated for RBHT experiment 1096.
In Table 4-3, the diameters, velocities and Weber numbers for the drops which have Weber numbers higher than 150.0 are highlighted. As the table indicates the smallest diameter for the large and intermediate size drops is approximately 0.00225 ft. Table 4-3 and further analysis using RBHT showed that, when drop diameter is 0.00221 ft, the Weber number becomes 150.0. Therefore, 0.00221 ft is the upper diameter limit for the small droplets, and it is used for the tabulated results for Kataoka et al. correlation using $D/D_{vm}$.

<table>
<thead>
<tr>
<th>Diameter (ft)</th>
<th>Velocity (ft/s)</th>
<th>Weber Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.001817346</td>
<td>9.77205</td>
<td>86.70</td>
</tr>
<tr>
<td>0.001856718</td>
<td>9.962686</td>
<td>92.07</td>
</tr>
<tr>
<td>0.001896746</td>
<td>10.15499</td>
<td>97.72</td>
</tr>
<tr>
<td>0.001937759</td>
<td>10.34886</td>
<td>103.68</td>
</tr>
<tr>
<td>0.001979755</td>
<td>10.54931</td>
<td>110.07</td>
</tr>
<tr>
<td>0.002022737</td>
<td>10.75155</td>
<td>116.81</td>
</tr>
<tr>
<td>0.002066374</td>
<td>10.95301</td>
<td>123.85</td>
</tr>
<tr>
<td>0.002110995</td>
<td>11.15925</td>
<td>131.33</td>
</tr>
<tr>
<td>0.002156601</td>
<td>11.37077</td>
<td>139.30</td>
</tr>
<tr>
<td>0.00220352</td>
<td>11.58489</td>
<td>147.75</td>
</tr>
<tr>
<td>0.002251094</td>
<td>11.7982</td>
<td>156.55</td>
</tr>
<tr>
<td>0.002299653</td>
<td>12.01609</td>
<td>165.88</td>
</tr>
<tr>
<td>0.002349524</td>
<td>12.23819</td>
<td>175.81</td>
</tr>
<tr>
<td>0.00240038</td>
<td>12.46536</td>
<td>186.34</td>
</tr>
<tr>
<td>0.002452219</td>
<td>12.6932</td>
<td>197.39</td>
</tr>
<tr>
<td>0.002505372</td>
<td>12.92298</td>
<td>209.03</td>
</tr>
<tr>
<td>0.002559508</td>
<td>13.15707</td>
<td>221.36</td>
</tr>
<tr>
<td>0.002614629</td>
<td>13.39366</td>
<td>234.33</td>
</tr>
<tr>
<td>0.00267139</td>
<td>13.63763</td>
<td>248.22</td>
</tr>
<tr>
<td>0.002729136</td>
<td>13.88216</td>
<td>262.76</td>
</tr>
<tr>
<td>0.002788194</td>
<td>14.13042</td>
<td>278.14</td>
</tr>
</tbody>
</table>

Table 4-3: Sample Results of Relative Velocity and Weber Number Calculations
For the COBRA-TF calculations, the tabulated volume fractions for small droplet field with different $D/D_{sm}$ ratios are plotted and a $5^{th}$ order polynomial is fit to this plot. Figure 4-1 presents these results and the curve fit. The polynomial obtained as the result of the curve fit is used as a function in COBRA-TF to determine the volume ratio of entrained small droplets for a calculated Sauter mean diameter and a given limit diameter of $D$.

Once the volume ratio of the entrained small droplets is determined, it is multiplied with the entrainment rate determined by using Holowach model. Finally it is used as a source term in the conservation of mass equation written for the small droplet field ($S_{sm}^*$ term in Equation (4-1)).
4.2.4 Large Drop Break up due to Aerodynamic Forces

As mentioned in Chapter 2, in series of experiments performed for dispersed flow, two types of break up mechanisms were observed:

Aerodynamic (bag type) break up: This type of break up occurs when the Weber number calculated using relative drop velocity is low. When the large drop deforms it takes a bag shape and small droplets are generated at the center of this bag shaped large drop. Figure 4-2 drawn by Ardron and Hall [45] illustrates this type of break up.

![Figure 4-2: Bag Type (Aerodynamic) Break up [45]](image)

The critical Weber number for aerodynamic break up is taken as 12.0 [8]. The Weber number was calculated by using Equation (4-23) which uses the relative velocity between the vapor and the large droplet.

\[
We = \frac{\rho_i(U_v - U_d)^2 x D_{drop}}{\sigma} \quad (4-23)
\]
If the Weber number is greater than 12.0, then the break up time \(t_1\) for the drop is calculated and it is compared with the time that a large drop would spend in the computational cell \(t_2\):

\[
t_1 = 6.5 \frac{D_o}{u_{ro}} \sqrt{\frac{\rho_l}{\rho_g}}
\]  \hspace{1cm} (4-24)

\[
t_2 = \frac{U_d}{\Delta x_{cell}}
\]  \hspace{1cm} (4-25)

Equation (4-24) is taken from Ardron and Hall [45] and for the equation constant, in order to be conservative, the upper limit they suggested for the constant value, i.e. 6.5, is used. In Equation (4-25), \(U_d\) is the drop velocity and \(\Delta x_{cell}\) is the computational cell length. If \(t_2\) is less than \(t_1\), large drop passes the cell before breaking into small droplets. If this is not true, the drop breaks up.

Once the break up occurs, the amount of the large drop mass that would be transformed to the small drops can be found using the upper limit log normal distribution and Figure 4-3.

Figure 4-3 shows summarized results of several researches for the variation of maximum residual droplet diameter with Weber number [48]. Figure 4-3 shows that, the ratio of the diameter of the largest droplet generated and the diameter of the broken large drop \(a_{fr}\) is approximately 0.2 ([48], [8]). Using this ratio and upper limit log normal distribution, The Sauter Mean Diameter (SMD) for the generated droplets can be found:

\[
SMD_{bagtype} = \frac{1}{3} a_{fr} D_{drop}
\]  \hspace{1cm} (4-26)

where \(D_{drop}\) is the diameter of the broken drop and \(a_{fr}D_{drop}\) is the diameter of the largest generated droplet.
Once the SMD is known, by using the volume ratio of the broken and generated drops, the large drop mass that is transformed to the small drop mass is found as:

\[
\frac{m_{\text{small}}}{m_{\text{large}}} = \frac{\pi}{6} \left( \frac{\text{SMD}_{\text{bagtype}}}{3} \right)^3 = \frac{1}{3} a_{fr} \left( \frac{D_{\text{drop}}}{3} \right)^3 = \left( \frac{1}{3} a_{fr} \right)^3
\]  

(4-27)

Since \( a_{fr} = 0.2 \) then;

\[
\frac{m_{\text{small}}}{m_{\text{large}}} \approx 0.0003 
\]  

(4-28)

In COBRA-TF, the mass flux for the large droplets is calculated. The small droplet mass flux is calculated using the ratio given in Equation (4-28).
Capillary (stripping type) break up: This break up occurs when the Weber number is higher, and when the shear on the drop surface is at a rate much faster than the rate at which drop inertia lets the drop deform as a single drop. Capillary break up is illustrated in Figure 4-4.

For the stripping type break up, the critical Weber number is set as 50.0 by Sarjeant [48]. When the Weber number for the large drop is greater than 50.0, break up occurs. Wolfe and Anderson [73] and Sarjeant [48] set the break up time for the stripping type break up as below:

$$t_1 = \frac{D_o}{u_{ro}} \sqrt{\frac{\rho_i}{\rho_g}}$$

(4-29)
The time that a drop would spend in a computational cell is calculated using Equation (4-25) and if it is greater than $t_1$, break up occurs. By using Figure 4-3 one can conclude that, the diameter ratio for the broken drop and largest droplet diameter is approximately 0.1 when the Weber number is 50.0 or larger (Fyall and King [74] and Sarjeant [48]). Therefore, the ratio of large drop mass shattered into small droplets can be found using the upper limited log normal distribution as described earlier:

$$SMD_{stripping} = \frac{1}{3} a_f D_{drop}$$  \hspace{1cm} (4-30)$$

$$\frac{m_{small}}{m_{large}} = \frac{\frac{\pi}{6} (SMD_{stripping})^3}{\frac{\pi}{6} (D_{drop})^3} = \frac{\frac{1}{3} a_f D_{drop}^3}{(D_{drop})^3} = \frac{1}{3} a_f^3$$  \hspace{1cm} (4-31)$$

Since $a_f = 0.1$ then;

Figure 4-4: Shear (Capillary) Break up [45].
\[
\frac{m_{\text{small}}}{m_{\text{large}}} \approx 0.000037 \tag{4-32}
\]

Although it is very rare that the drops would break up due to aerodynamic forces during reflood, and although the mass generated for small droplet field is very small, since an additional interfacial area for vapor and small droplet fields is generated, in the present study, drop break up due to aerodynamic forces are taken into account.

## 4.2.5 Interfacial Drag Models

Interfacial drag models are used to model the interfacial momentum exchange between the phases. In COBRA-TF, these models are flow regime dependent.

The models developed for the interfacial drag depend on the relative velocity between the phases. These models also depend on interfacial area, volume fraction of the phases and mass flux in a computational cell.

In COBRA-TF, for dispersed droplet film boiling, the drag model developed by Ishii [75] for a single bubble in an infinite liquid field is used using the analogy of a single drop in an infinite vapor. The drag model used for the vertical flow is:

\[
\tau = \frac{0.125 C_d A_{\text{drop}} \rho_v |U_d|}{\Delta X} \tag{4-33}
\]

In Equation (4-33), \( A_{\text{drop}} \) is the interfacial area between the drop and the vapor, \( \rho_v \) is the vapor density, \( |U_d| \) is the relative velocity between the drop and vapor field and \( \Delta X \) is the computational cell length. In drag model \( C_d \) is the drag coefficient and in COBRA-TF for lateral velocity it is given by:

\[
C_{d_{\perp}} = \frac{24}{\text{Re}_{D}} (1 + 0.1 \text{Re}_{D}^{0.75}) \tag{4-34}
\]
In above relation \( \text{Re}_D \) is the Reynolds number of the drop field and it is given by Equation (4-21).

COBRA-TF uses a constant drag coefficient for lateral flow:

\[
C_d = 0.45
\]  
(4-35)

Interfacial drag for lateral flow is calculated as:

\[
\tau = 0.375C_d \rho_v |\dot{W}_d| \alpha_e L_g \Delta Z / r_d
\]  
(4-36)

where \( |\dot{W}_d| \) is the relative lateral velocity for the drop field, \( \alpha_e \) is the volume fraction for the entrained drops, \( L_g \Delta Z \) is the lateral surface area for the computational cell and \( r_d \) is the drop radius.

Equations (4-33) through (4-36) present that, the drag models currently used in COBRA-TF depend on drop size, interfacial area, drop volume fraction and relative velocity between drop and the vapor. Since all of these parameters are calculated for the small droplet field by solving the conservation equations and interfacial area transport equation, same interfacial drag models can be used for the large drop and small droplet fields.

### 4.2.6 De-entrainment in Liquid Film/Drop Flow

Liquid film can exist during liquid film/drop flow which is a cold wall flow regime. On the ends of quenched rods, liquid film exists and the deposition of drops on this film is possible.

The deposition of the drops on this liquid film occurs due to turbulent motion of the flow which pushes droplets through the liquid film. In COBRA-TF, the rate of de-entrainment is determined as [60]
\[ S_{DE} = k_s \times \Delta C \times P_w \times \Delta X \]  \hspace{1cm} (4-37)

where, \( k_s \) is the mass transfer coefficient, \( P_w \) is the wetted perimeter and \( \Delta X \) is the cell height. In Equation (4-37) \( \Delta C \) is the drop concentration gradients for the channel:

\[ \Delta C = \frac{\alpha_{sm} \rho_f}{\alpha_{sm} + \alpha_e + \alpha_v} \]  \hspace{1cm} (4-38)

The mass transfer coefficient is calculated as a function of surface tension:

\[ k_s = \max(3.0492 \times 10^{12} \times \sigma^{5.3054}, 12.491 \times \sigma^{0.8968}) \]

### 4.2.7 Interfacial Area Transport Equations

The interfacial area transport equation and COBRA-TF solution methodology for the equation for the entrained drop field is described in Chapter 3. Interfacial area transport equation is:

\[ \frac{dA_{i,d}}{dt} + \nabla \cdot (A_{i,d} U_e) = A_{i,E}^{\pi} + A_{i,G}^{\pi} \]  \hspace{1cm} (3-120)

For the five field model, for large drop and small droplet fields, similar equations are solved. The area transport equation for large drop field is:

\[ \frac{dA_{i,d}^{\pi}}{dt} + \nabla \cdot (A_{i,d}^{\pi} U_e) = A_{i,E}^{\pi} + A_{i,G}^{\pi} + A_{BDS,LD}^{\pi} + A_{BDF,LD}^{\pi} \]  \hspace{1cm} (4-39)

In Equation (4-39), \( A_{i,E}^{\pi} \) is the area change due to entrainment, \( A_{i,G}^{\pi} \) represents the change due to evaporation and the model used for \( A_{i,G}^{\pi} \) is described in Section 3.7. The change in interfacial area due to convection is represented by \( \nabla \cdot (A_{i,d}^{\pi} U_e) \) term.
The new terms introduced with the five field model are $A_{BDS,LD}^{\text{LD}}$ and $A_{BDF,LD}^{\text{LD}}$. These terms represent the change in interfacial area for the large drop field due to break up at the spacer grids and break up due to forces, respectively.

For the small droplet field a separate area transport equation which is similar to Equation (4-39) can be written:

$$
\frac{dA_{SM}^{\text{SM}}}{dt} + \nabla \cdot (A_{SM}^{\text{SM}} \mathbf{U}_{SM}) = A_{i, \Gamma, SM}^{\text{SM}} + A_{BDS, SM}^{\text{SM}} + A_{BDF, SM}^{\text{SM}} + A_{i, E, DE}^{\text{SM}} \tag{4-40}
$$

To solve above equation, to calculate the change in area due to evaporation, the same models used for the large drop field are also used for the small droplets. The change in small droplet interfacial area due to evaporation becomes:

$$
A_{i, \Gamma, SM}^{\text{SM}} = A_{i, \Gamma, SM}^{\text{SM}} \left( 1 - \frac{\eta_{SM} \Gamma_{SM}}{\rho \alpha_{SM} V_c} \right)^{2/3} \tag{4-41}
$$

where $A_{i, SM}^{\text{SM}}$ is the interfacial area at the old time step. In above equation $\eta_{SM} \Gamma_{SM}$ is the amount of vapor generated due to evaporation of small droplets.

To calculate the changes due to entrainment and break up, COBRA-TF uses the mass generated as the result of these mass transfer mechanisms. The models used for the mass generation due to entrainment and break up are described in Sections 3.6.3, 4.2.3 and 4.2.4. Once the mass generation parameter is calculated, COBRA-TF calculates the change in interfacial area due to entrainment using the relationship below as:

$$
A_{i, E, SM}^{\text{SM}} = \frac{S_{SM}^{\text{SM}}}{D_{E, SM}} \tag{4-42}
$$

where $D_{E, SM}$ is the droplet formation size. COBRA-TF evaluates the droplet formation size as:
where, \( EVR \) is the volume ratio for the small droplets generated by the entrainment described in Section 4.2.3. In Equation (4-43), \( D_H \) is the hydraulic diameter of the channel and \( D_1 \) and \( D_2 \) are the drop diameters evaluated using experimental correlations depending on the fluid properties:

\[
D_1 = \frac{1.13}{((\rho_l - \rho_v)/\rho_v)^{0.865}} \tag{4-44}
\]

\[
D_2 = \frac{0.3 \sigma}{\rho_l - \rho_v} \tag{4-45}
\]

In above equation, \( \sigma \) is the surface tension of the liquid. The Equations (4-44) and (4-45) were derived using FLECHT data [59].

The change in area due to break up at the spacer grid or flow blockages is evaluated using a similar relation to the Equation (4-42)

\[
A_{BDS,SM} = \frac{S_{BDS}}{SMD^n} \tag{4-46}
\]

where \( SMD^\circ \) is the Sauter mean diameter evaluated using new time volume fraction and old time area density.

The change in interfacial area because of the break up due to aerodynamic forces becomes:

\[
A_{BDS,SM} = \frac{S_{BDF}}{SMD^\circ} \tag{4-47}
\]

The effect of de-entrainment on the change in interfacial area is calculated similarly:
Once all the corresponding terms in the interfacial area transport equations are calculated, the equations are solved by applying the method described in Section 3.7.

### 4.3 Numerical Solution Methodology for Five Field Model

Once all the closure relationships are evaluated and interfacial area between the fields is determined, the set of new and modified conservation equations can be solved.

As mentioned earlier, two momentum equations (axial and transverse) and one mass equation are added to the COBRA-TF solution scheme for a five field model. The increase in number of equations to be solved increases the computation time for the problem in hand. For a problem with large number of meshes, solving three more equations for each mesh in the model would be computationally harder. In order to prevent computation from being very long, several programming and numerical techniques are adopted for the five field model.

In order to minimize the computation time, using five field model is made optional to the user. Since modeling the small droplet field is important during the blowdown and reflood phases of a loss of coolant accident, for a problem which does not simulate these phases, solving extra equations for small droplet field can be computationally expensive. Therefore, if a COBRA-TF user wants to simulate the small droplet field, he or she can easily change one variable in the input deck and have a five field flow simulation.

As mentioned in Chapter 3, for the numerical solutions, momentum equations are solved first to obtain the tentative flow rates and velocities for the fields. The momentum equations with the five field model are:

\[
A_{BDS,SM}^{\alpha} = \frac{S_{DE}^{\alpha}}{SMD^3}
\]  

(4-48)
\[ F_l = A_1 + B_1 \Delta P + C_1 F_i + D_1 F_v \text{ (Liquid)} \] (4-49)

\[ F_v = A_2 + B_2 \Delta P + C_2 F_i + D_2 F_v + E_2 F_e + G_2 F_{sm} \text{ (Vapor)} \] (4-50)

\[ F_e = A_3 + B_3 \Delta P + D_3 F_v + E_3 F_e \text{ (Large Drop)} \] (4-51)

\[ F_{sm} = A_4 + B_4 \Delta P + D_4 F_v + E_4 F_e + G_4 F_{sm} \text{ (Small Droplet)} \] (4-52)

In above equations there are several new terms. For example, \( A_i \) is the constant for the explicit term in the momentum equation, \( B_i \) is the explicit portion of the pressure gradient source term, \( D_i, E_i, G_i \) and \( G_i \) are the factors that multiply the vapor, large drop and small droplet flow rates with the wall and interfacial drag terms.

The new matrix of the momentum equations is:

\[
\begin{bmatrix}
C_1 - 1 & D_1 & 0 & 0 \\
C_2 & D_2 - 1 & E_2 & G_2 \\
0 & D_3 & E_3 - 1 & 0 \\
0 & D_4 & E_4 & G_4 - 1
\end{bmatrix}
\begin{bmatrix}
F_l \\
F_v \\
F_e \\
F_{sm}
\end{bmatrix} =
\begin{bmatrix}
-A_1 - B_1 \Delta P \\
-A_2 - B_3 \Delta P \\
-A_3 - B_3 \Delta P \\
-A_4 - B_3 \Delta P
\end{bmatrix}
\] (4-53)

In order to solve this 4 x 4 matrix system, a Gaussian elimination with scaled partial pivoting method is used [76]. This method uses a scale vector in order to determine a pivot row. For naive Gaussian elimination used in COBRA-TF, the pivot row is always the first row of the matrix. Using a different pivot row other than the first row improves the accuracy of the elimination.

In order to solve the mass and energy equations using the solution for the flow rates from this elimination, the same solution technique used in COBRA-TF is introduced. Because of its very complicated structure, the Jacobian matrix given by Equation (3-73) is not changed to add the terms for the mass equation of the small droplet...
field. Instead, the new time values of the volume fraction of the small droplet field is obtained by using a separate relation once the pressure problem is solved as described in Chapter 3. Williams [57] used this technique to calculate the new time value for the volume fraction of the two drop fields he added to the two fluid solution system. Since a thermal equilibrium between the fields is assumed, the continuous liquid, large drop and small droplet fields are thermodynamically same for the problem solution. Therefore, small droplet field mass equation can be solved separately to obtain the volume, i.e. the mass fraction for this field.

The new time volume fraction for the small droplet field can be calculated as below:

\[
\left(\alpha_{s,m}\right)_{j} = \frac{1}{\rho_{ij}} \left(\left(\alpha_{s,m}\rho_{t}\right)_{j}^{n}\right) + \frac{\Delta t}{A_{ij}} \left[ \sum_{K=1}^{N} \left(\alpha_{s,m}\rho_{t}\right)_{j}^{n} U_{s,m,i} A_{m,j-1}^{n} \right]_{KB} \left[ \sum_{K=1}^{N} \left(\alpha_{s,m}\rho_{t}\right)_{j}^{n} U_{s,m,j} A_{m,j}^{n} \right]_{KA} \\
+ \sum_{K=1}^{N} \sum_{L=K+1}^{M} \left(\alpha_{s,m}\rho_{t}\right)_{j}^{n} W_{s,m,j} L_{A} \left( \frac{\Gamma_{s,m}^{n}}{\Delta X_{j}} + \frac{S_{BDS}^{n}}{\Delta X_{j}} + \frac{S_{BDF}^{n}}{\Delta X_{j}} \right)
\]

(4-54)

Applying this technique to COBRA-TF requires adding new arrays to store the variables for old time and new time values of all the variables in Equation (4-54) and the Equation (3-51) used for the large drop field. Adding large numbers of arrays for each computational cell is also computationally expensive and can cause programming errors and problems easily. Therefore, small droplet mass equation is added to Jacobian matrix and solution set is defined as below:
Once the Jacobian matrix is defined as above, same calculations are performed as described in Chapter 3 to solve 5 mass and 2 energy equations for the five fields.

\[ \begin{bmatrix}
\frac{\partial E_{cg}}{\partial \alpha_g}, \frac{\partial E_{cg}}{\partial \alpha_v}, \frac{\partial E_{cg}}{\partial \alpha_{H_v}}, \frac{\partial E_{cg}}{\partial (1-\alpha_v)H_I}, \frac{\partial E_{cg}}{\partial \alpha_e}, \frac{\partial E_{cg}}{\partial \alpha_{sm}}, \frac{\partial E_{cg}}{\partial P_j}, \frac{\partial E_{cg}}{\partial P_i=NCON}
\end{bmatrix} =
\begin{bmatrix}
\frac{d\alpha_g}{d\alpha_v}, \frac{d\alpha_g}{d\alpha_e}, \frac{d\alpha_g}{d(1-\alpha_v)H_I}, \frac{d\alpha_g}{d\alpha_e}, \frac{d\alpha_g}{d\alpha_{sm}}, \frac{d\alpha_g}{dP_j}, \frac{d\alpha_g}{dP_i=NCON},
\end{bmatrix}
\]

(4-55)

**4.4 Modifications to Spacer Grid Models**

The spacer grid models used in COBRA-TF are described in Chapter 3. One of the most important spacer grid models for the two phase dispersed drop film flow is the drop break up model and the current model is used as a mass source for the small droplet field without any modification in the five field model.

Rod Bundle Heat Transfer project experiments show that, grid rewet is also important in modeling the effects of spacer grids on heat transfer. Figure 4-5 presents a sample measurement for grid, steam and rod surface temperatures. As Figure 4-5 shows, the grid quenches much earlier than the rods and the steam probes. Once the spacer grid is quenched and when there is a liquid film formed on it, a large heat transfer area
between the colder wet spacer grids and superheated vapor is generated. For an accurate modeling of heat transfer, effects of wet spacer grid must be taken into account.

The original COBRA-TF calculates the interfacial heat transfer area between the liquid film on the grid and vapor. This area is used to calculate the energy transfer to the vapor, but code does not take the amount of vapor mass generated at saturation temperature and momentum transferred into account.

The initial temperatures of the spacer grids are estimated as high as the rod temperatures by the original code. As Figure 4-5 presents, the initial temperature of the spacer grid is much lower than both the steam and the heater rod temperatures.

Most importantly, in order to simulate the grid quench, the only liquid mass the original code is using is the de-entrainment rate of the drops. The original code does not take into account the amount of liquid mass deposited on the spacer grid, or mass loss from this deposited liquid due to evaporation or the entrainment.

Figure 4-6 presents the spacer grid temperature calculated for one of the RBHT experiments calculated by the original code. As Figure 4-6 indicates, the grid quench is not simulated correctly by the code. This causes a higher estimation of large drop break up since for the quenched grid drop break up is not simulated. In addition, a smaller vapor generation from the liquid film is estimated due to inaccurate modeling of grid quench.

In order to obtain a better simulation for the spacer grids, the code was modified to read the initial grid temperatures from the input deck. Therefore the initial temperatures for the spacer grid must be entered by the user. In addition, for the liquid film on the spacer grids, mass and momentum equations are solved. Solving these equations is helpful to simulate grid quench, evaporation from the liquid film, the vapor drag on the film and drop deposition and entrainment. At each time step, below mass equation is solved:
\[
\frac{[SGM -(SGM)'']}{\Delta t} = \Gamma_{\text{grid}} + S_{DEgrid} - S_{Egrid}
\]

(4-56)

In Equation (4-56) \(\Gamma_{\text{grid}}\) is the evaporation from the liquid film on the spacer grid, \(S_{DEgrid}\) is the de-entrainment rate on the grids and \(S_{Egrid}\) is the entrainment rate from the spacer grid liquid film.

The evaporation from the liquid film on the spacer grid \(\Gamma_{\text{grid}}\), is calculated by taking the heat transferred to the liquid film into account. In original COBRA-TF, heat transfer to wet portion of the grid is calculated:

\[
HT_{\text{wet,grid}} = HA_g Q_F (T_{\text{vap}} - T_{\text{sat}}) + (q_{\text{rad,\text{w}}} P W_{\text{grid}} Q F_{\text{grid}}) \times \text{no. of grids}
\]

(4-57)
where, \( q_{\text{rad,w}} \) is the radiation heat transfer to the wet portion of the grid, as described in Section 3.6.2 and \( HA_{\text{eq}F} \) is the interfacial heat transfer coefficient calculated for the heat transfer from the liquid film to two phase vapor drop mixture:

\[
HA_{\text{eq}F} = (TP_{\text{enh}}k_{\text{film}}, Nu_{\text{grid}}PW_{\text{grid}}Q_{\text{Fgrid}}) \times \text{no. of grids}
\]  

(4-58)

Figure 4-6: Comparison of the Grid Temperature Measurement and the Grid Temperature Calculated with the Original COBRA-TF. (RBHT Experiment 1096 (20 psia, 1.0 in./sec Inlet Flooding Rate, 20 °F Inlet Subcooling))

In above equations, \( PW_{\text{grid}} \) is the wetted perimeter for the grids, \( Q_{\text{Fgrid}} \) is the length of the wetted portion of the grid, \( k_{\text{film}} \) is the thermal conductivity of the vapor film and \( TP_{\text{enh}} \) is the two phase enhancement of the vapor heat transfer coefficient as described in Section 3.4. In Equation (4-58) the Nusselt number for the grid is calculated as:

\[
Nu_{\text{grid}} = \max(10, 0.0797 \times Re^0.6774 \times Pr^{0.333}) \text{ (if } Re_v < 25200) 
\]  

(4-59)
\[ Nu_{grid} = 0.023 \times Re_v^{0.8} \times Pr_v^{0.4} \quad (if \ Re_v > 25200) \] (4-60)

where \( Re_v \) is the Reynolds number and \( Pr_v \) is the Prandtl number calculated for the vapor.

Using the heat transfer relation calculated by using Equation (4-57), evaporation from the liquid film on the grid is calculated as:

\[ \Gamma_{grid} = \frac{HT_{wet,grid}}{h_{fg}} \] (4-61)

where \( h_{fg} \) is the heat of vaporization.

The de-entrainment on the spacer grids is calculated as:

\[ S_{DEgrid} = \frac{A_g}{A_c} \dot{m}_E \] (4-62)

where \( A_g \) is the grid projected area, \( A_c \) is the channel flow area and \( \dot{m}_E \) is the entrained liquid rate in the channel.

In order to calculate the entrainment from the spacer grid liquid film, \( S_{Egrid} \), Holowach’s model [70] for the film entrainment is applied.

The momentum equation solved for the liquid film on the grids is:

\[
\begin{aligned}
\left[ (SGMom) - (SGMom)^n \right] &= -\frac{U_{grid}n \rho \bar{A}_g K_{grid,l}}{\Delta X_j} + \frac{U_{v}n \rho \bar{A}_c K_{v,grid}}{\Delta X_j} \\
\Gamma_{grid} \frac{U_{grid}n}{\Delta X_j} + (S_{DEgrid}U_{gr}n - S_{Egrid}U_{grid}n) &\quad (4-57)
\end{aligned}
\]
where \( U_{\text{grid}} \) is the velocity of the spacer grid liquid film calculated using channel flow area, liquid film flow rate calculated by using Equation (4-57) and liquid density:

\[
U_{\text{grid}}^n = \frac{SGMom}{\rho_l A_g} \tag{4-58}
\]

In Equation (4-57) \( U_v^n \) and \( U_e^n \) are the old time values of velocities of vapor and entrained liquid phases, respectively.

In the momentum equation solved for the liquid film on the spacer grid, \( K_{\text{grid}, i} \) is the wall drag coefficient calculated as:

\[
K_{\text{grid}, i} = f_i \rho_l U_{\text{grid}}^2 \frac{h_{\text{grid}, i}}{2D_h} \tag{4-59}
\]

where \( D_h \) is the hydraulic diameter and \( f_i \) is the friction factor for the liquid. Friction factor is calculated as below:

\[
f_i = \frac{64}{Re} \text{ (laminar)} \tag{4-60}
\]

\[
f_i = 0.0055 + 0.55 Re_i^{1/3} \text{ (turbulent)} \tag{4-61}
\]

The other drag coefficient in Equation (4-57) is the interfacial drag coefficient that must be evaluated for vapor and liquid film, \( K_{i, \text{grid} i} \):

\[
K_{i, \text{grid} i} = f_{i, \text{grid} i} \rho_v (U_v - U_{\text{grid} i}) (\alpha_{\text{grid}, i})^{0.5} QF_{\text{grid}} \tag{4-62}
\]

where \( \alpha_{\text{grid}, i} = \frac{SGMom}{SGMom + U_v \rho_v A_v + U_e \rho_e A_e} \) and represents the volume ratio of liquid film in the channel. In Equation (4-62) \( f_{i, \text{grid} i} \) is the interfacial drag coefficient calculated as [78]:
The evaporation rate, entrainment and de-entrainment calculated for the spacer grids are coupled with code’s solution scheme as source or loss terms for the mass, momentum and energy equations as long as the heat transfer regime is a hot wall flow regime. For the cold wall flow regimes like film/drop regime, since liquid film is already exist on both the rods and the spacer grids, mass, momentum and energy transfer terms calculated for the spacer grid liquid film are not taken in to account in the solution scheme.

Once the amount of liquid mass deposited on the spacer grid is determined, the change in quench front location is determined. In COBRA-TF, quench front progress is limited by the availability of the liquid and heat conduction rate between the wet and dry regions of the grid.

When as the result of the solution of the mass equation, if the liquid amount and drop deposition rate is greater than the evaporation and entrainment rate, quench front may progress. The quenching of a thin plate by liquid film is modeled by Yamanouchi and this model is used in COBRA-TF [60]. The velocity of quench front, \( V_Q \), is determined as:

\[
V_Q = \left[ \frac{\rho_g C_{pg}}{2} \left( \frac{\delta}{h_w k_g} \right)^{1/2} \left( 1 + \frac{T_g - T_w}{T_w - T_l} \right) \left( 1 + 2 \frac{T_g - T_w}{T_w - T_l} \right)^{1/2} - 1 \right]^{-1}
\]  

(4-64)

where, \( \rho_g \) is the density, \( C_{pg} \) is the specific heat and \( k_g \) is the thermal conductivity of the grid material. In Equation (4-64), \( \delta \) is the grid half thickness and \( T_g \) and \( T_w \) are the dry region and wet region temperatures, respectively.

In Equation (4-64) \( h_w \) is the wet region heat transfer coefficients and in COBRA-TF it is set to:
where, \( q_{CHF}^{*} \) is the critical heat flux determined by using Zuber pool boiling critical heat flux correlation [66].

The available water for the quenching is determined by taking mass stored on the grid, evaporation, drop deposition and entrainment rates:

\[
\dot{m}_R = (SGM)^n - \Gamma_{grid} + S_{DEgrid} - S_{Egrid}
\]  

(4-66)

The amount of stored energy removed at the quenched front is calculated using an efficiency factor to take the fraction of water that can be evaporated:

\[
ENG_{stored} = \dot{m}_R \times \exp \left[ 1 - \left( \frac{T_g}{T_{sat}} \right) \right]^2 \times h_{fg}
\]  

(4-67)

The energy that would be removed at the quench front can not exceed this stored energy, and Equation (4-67) can be used to limit the quench front velocity:

\[
ENG_{stored} \geq (\rho C_p A_g) \times V_Q \times (T_g - T_{sat})
\]  

(4-68)

therefore,

\[
V_Q \geq \frac{ENG_{stored}}{(\rho C_p A_g) \times V_Q \times (T_g - T_{sat})}
\]  

(4-69)

Using Equations (4-64) and (4-69) a limit for wet region heat transfer is derived:

\[
h_w \leq \left( \frac{\delta}{4k_g} \right) \left( \frac{ENG_{stored}}{P_g \delta (T_g - T_w)} \right)^2 \left( \left( 1 + \frac{T_g - T_w}{T_w - T_l} \right)^2 - 1 \right)
\]  

(4-70)
where, $P_g$ is the wetted perimeter of the grid.

The heat transfer coefficient that can be used to calculate the quench front velocity is selected as the maximum of the heat transfer coefficients determined by using Equations (4-65) and (4-70).

### 4.5 Modifications to Model for Convective Enhancement Due to Droplets

As described earlier in Section 3.4 due to interfacial shear between the continuous vapor and droplets, turbulence in vapor flow increases. The increase in turbulence causes convective enhancement in dispersed flow film boiling.

The model used in original COBRA-TF was derived using the results of reflood data obtained during FLECHT Cosine, FLECHT Skewed and FLECHT-SEASET experiments. Figure 4-7 presents the comparison of models and these experimental data [60].

As Figure 4-7 indicates when the $(1-\alpha_v)Gr/Re^2$ term is less than 0.0001, there is not experimental data that would prove the agreement with the original COBRA-TF model. In order to prevent the over prediction of convective enhancement and the over prediction of two-phase heat transfer coefficient, the convective enhancement model was modified as:

$$f_{2q} = \left(1 + \frac{\tau_d}{\tau_w}\right)^{1/2}, \text{ (when } (1-\alpha_v)Gr/Re^2 > 0.0001) \quad (4-71)$$

where as explained in section 3.4:

$$\frac{\tau_d}{\tau_w} = 1.5\alpha_{e}\left(\frac{D_H}{D_d}\right)\left(\frac{C_{D_d}D_d}{f_w}\right)\left(\frac{U_v - U_d}{U_v}\right)^2 \quad (4-72)$$
\[ f_{20} = 1.0 \quad \text{(when} \quad (1 - \alpha_v)Gr / Re^2 < 0.0001) \quad (4-73) \]

Here, \( Gr \) is the Grashof number (ratio of buoyancy forces to the viscous forces) calculated for vapor phase:

\[ Gr = \frac{g \beta |T_w - T_r| D_{h}^2 \rho_v \beta}{\mu_v^2} \quad (4-74) \]

where \( g \) is the gravitational acceleration, \( \beta \) is the thermal expansion coefficient, \( T_w \) and \( T_r \) are the temperatures of rods and vapor, respectively. In Equation (4-74) \( \mu_v \) is the dynamic viscosity of the vapor.

### 4.6 Modifications to Minimum Film Boiling Temperature Model

In order to model heat transfer in reflood phase of a postulated LOCA accurately, an accurate calculation of minimum film boiling temperature (\( T_{\text{min}} \)) is very important. In COBRA-TF, transition from film boiling to transition boiling is determined by comparing the rod temperature with the minimum film boiling temperature calculated for a computational cell.
Earlier studies performed to develop $T_{\text{min}}$ correlations show that $T_{\text{min}}$ is a function of pressure, subcooling and the properties of the surface material. In the original COBRA-TF code, the minimum value for $T_{\text{min}}$ is set to 482.2 °C (900 °F), and the maximum $T_{\text{min}}$ value is 648.9 °C (1200 °F). Other than these empirically determined values, COBRA-TF uses the wall temperature which is equal to homogeneous nucleation temperature $T_{hn}$ to calculate $T_{\text{min}}$:
where $T_{hn}$ is a function of pressure:

$$T_{hn} = 705.44 - 4.722 \times 10^{-2} \Delta P_{\text{crit}} + 2.3907 \times 10^{-5} \Delta P_{\text{crit}}^2 - 5.8193 \times 10^{-9} \Delta P_{\text{crit}}^3$$  \hspace{1cm} (4-76)$$

In Equation (4-69), $\Delta P_{\text{crit}} = 3203.6 - P$ and $P$ is the pressure in psi.

Recently, Peterson and Bajorek [79] developed a new $T_{\text{min}}$ model using the experimental data from high pressure quenching experiments performed for vertical cylinders. The new model takes surface roughness into account to estimate the $T_{\text{min}}$:

$$T_{\text{min}} = T_i + 0.095(T_i - T_w) \left( \frac{R_a}{R_{\text{ref}}} \right)^{0.283} \left( \frac{k\rho C_p}{C_p} \right)_{l}^{0.170} \left( \frac{h_{fg}}{(C_p)_{w}(T_i - T_{\text{sat}})} \right)^{0.957}$$ \hspace{1cm} (4-77)$$

where,

$$T_i = \min(T_{\text{min}}^B, T_{HN})$$ \hspace{1cm} (4-78)$$

In Equation (4-78), $T_{\text{min}}^B$ is $R_t$ model developed by Berenson and $T_{HN}$ is expressed as:

$$T_{HN} = T_{CR} \left[ 0.905 + 0.095 \left( \frac{T_{\text{sat}} + 273.15}{T_{CR}} \right)^{0.8} \right]$$ \hspace{1cm} (4-79)$$

where $T_{CR}$ is the critical temperature. $T_{\text{min}}^B$ is a function of flow properties and calculated as:

$$T_{\text{min}}^B = T_{\text{sat}} + 0.127\rho_g \frac{h_{fg}}{k_g} \left[ \frac{g(\rho_f - \rho_g)}{\rho_f + \rho_g} \right]^{2/3} \left[ \frac{\sigma}{g(\rho_f - \rho_g)} \right]^{1/2} \left[ \frac{\mu_g}{g(\rho_f - \rho_g)} \right]^{1/3}$$ \hspace{1cm} (4-80)$$
where vapor properties are evaluated at the film temperature.

In Equation (4-70), \( R_a \) is the surface roughness and \( R_{ref} \) is the reference surface roughness of 1.0 \( \mu \text{m} \).

If surface roughness is unknown, Peterson and Bajorek suggest using below \( T_{min}^B \) correlation:

\[
T_{min} = T_i + 0.239(T_i - T_w)\left(\frac{(k \rho C_p)_l}{(k \rho C_p)_w}\right)^{0.250} \left(\frac{h_{fg}}{(C_p)_w(T_i - T_{sat})}\right)^{0.832}
\]

(4-81)

In order to modify the \( T_{min} \) model in COBRA-TF both Equations (4-77) and (4-81) are added to the code. For the surface roughness, surface roughness for the RBHT rod roughness is used for the analysis.
Chapter 5

REFLOOD EXPERIMENTS

The proposed model for dispersed flow film boiling (DFFB) simulation using COBRA-TF is described in Chapter 4 and in order to validate the model, the COBRA-TF predictions must be compared with the proper experimental data.

Since the dispersed flow film boiling during reflood phase is modeled by the current study, the reflood experiments and data from these experiments are useful to validate the COBRA-TF five field model calculations.

There are several experiments performed for reflood in tubes. The closure relationships used in the conservation equations are evaluated using the reflood in tube data. For example, aerodynamic and capillary drop break up models are evaluated by using reflood in tube experiments [45]. The tube experiments are performed without a flow blockage or spacer grids and the data of the tube experiments are useful to validate the DFFB models without any flow blockage and spacer grids. Some of the models used in the present study have been developed by several researchers using the data from tube experiments. These models are described in Chapter 4.

There are two sets of experiments which include reflood tests performed for the rod bundle arrays with spacer grids. These experiments are Full Length Emergency Core Heat Transfer-System Effects and Separate Effects Tests (FLECHT SEASET) and Rod Bundle Heat Transfer (RBHT) tests. Especially the data from FLECHT SEASET experiments are widely used for the model evaluation and validation for reflood by many researchers. Most of the COBRA-TF models described in Chapter 3 have been evaluated using FLECHT SEASET data.
The reflood data from recently performed RBHT experiments are also very useful to validate the model described in Chapter 4 since the RBHT reflood experiments were performed with a broad range of boundary and initial conditions. These experiments were especially designed to be used for code validation, particularly for determining the effects of spacer grids. In addition, the models proposed for the closure relationships were evaluated using the data from experiments other than RBHT data; therefore, RBHT data are very helpful to independently validate the COBRA-TF five field model predictions for reflood as a separate set of data.

In this chapter, the FLECHT SEASET and RBHT experimental facilities and selected reflood experiments are described. The experiments with low pressure, high initial temperature and low flow injection rates are selected since, only in these cases dispersed droplet film boiling exists for a long period of time.

In addition to the descriptions of the test set ups and experiments, the differences between the FLECHT-SEASET and RBHT experiments are listed at the end of this chapter.

5.1 FLECHT SEASET Experiments

FLECHT SEASET experiments are a series of experiments performed by Westinghouse/NRC and EPRI. The reflood experiments were performed for 161-rod unblocked bundle. The test facility is designed to perform either the forced flooding or the gravity flooding experiments. Heater rods have the same dimensions with full length pressurized water reactor bundles and they are heated internally with a chopped cosine axial power distribution with 1.6 peak to average ratio. The bundle has 8 spacer grids located 50.0 cm apart along the heater rods. These spacer grids are made of stainless steel.

Figure 5-1 presents the cross sectional view of the test bundle [42]. There are 68 instrumented, 93 non instrumented heater rods, 4 thimbles, 12 steam probes and 8 solid
triangular fillers. The fillers are used to fix the grid locations and are welded to the spacer grids, as well as to reduce the excess flow area such that the bundle has a typical power to flow characteristics as a 17x17 nuclear fuel assembly.

The test facility includes a lower plenum, an upper plenum, a steam water separator, liquid collector tank and water injection system. The test loop schematic is presented in Figure 5-2 [42]. The temperature limit for the heater rods is 1110 °C and the nominal pressure for the bundle is 413.69 kPa (60 psia).

The reflood tests performed during FLECHT SEASET program designed to provide a better understanding of heat transfer mechanisms during dispersed flow film boiling regime. In order to obtain detailed information, the vapor superheat was measured using aspirating steam probes. These steam probes are designed such that they separate moisture from the high temperature steam and they aspirate the steam across the

Figure 5-1: FLECHT SEASET Bundle Cross Section [42]
thermocouple. This provides the information to determine the energy split between superheating of the vapor and the droplet evaporation separately.

The droplet size, distribution and velocity were measured using high speed black and white movies.

Some of the reflood experiments performed during FLECHT SEASET program are described in following sections.

![Figure 5-2: Schematic of FLECHT-SEASET Test Loop [42]](image)

**5.1.1 FLECHT SEASET Run 31504**

FLECHT-SEASET Run 31504 was selected as the base experiment by many researchers for model evaluation and validation since it simulates the postulated reflood conditions for a typical PWR. The test was conducted in the 161 heated rod FLECHT facility with a uniform radial power profile chopped cosine axial power profile. The range of conditions for Run 31504 are listed in Table 5-1.
5.1.2 FLECHT-SEASET Run 31922

FLECHT-SEASET Run 31922 was conducted at both a lower pressure and a slightly higher injection flow rate than the run 31504. This run is a characteristic to a reflood transient in a lower-pressure containment design. In Table 5-2, the conditions for this run are listed.

Table 5-2: Conditions for FLECHT-SEASET Run 31922

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Upper Plenum Pressure</td>
<td>0.14 MPa</td>
</tr>
<tr>
<td>Initial Clad Temperature at 1.83m</td>
<td>1156 K</td>
</tr>
<tr>
<td>Rod Peak Power</td>
<td>1.3 kw/m</td>
</tr>
<tr>
<td>Cold Fill Rate</td>
<td>27.2 mm/sec</td>
</tr>
<tr>
<td>Injected Coolant Temperature</td>
<td>308K</td>
</tr>
</tbody>
</table>

5.2 RBHT Experiments

Rod Bundle Heat Transfer test set up was developed by the US Nuclear Regulatory Commission and the Penn State University to obtain needed and new data on heat transfer and two-phase flow behavior in rod bundles [77]. The Rod Bundle Heat Transfer Test Facility, which is located at University Park, PA, is one of the newest test facilities that has been constructed for the expressed purpose of improving the analytical modeling capability of the US Nuclear Regulatory Commission such that best-estimate thermal-hydraulic calculations can be made, in confidence, with reduced uncertainty.

Figure 5-3 shows a schematic of the test facility and indicates that it is a once-through flow facility in which either water or steam can enter the lower plenum and flow upward through the rod bundle. There are a small liquid carryover tank and a larger liquid carryover tank which are attached to the upper plenum which measure the entrained liquid flow which is carried out of the bundle by the steam. Separating the exit flows from the bundle provides a method to perform a transient mass balance as well as an energy balance on the facility. In order to damp out any pressure oscillations such that it is easier to maintain tight pressure control a pressure damping tank is used in the facility.
The facility has been designed to perform forced reflood tests, with liquid injection into the lower plenum, steam cooling experiments with steam injection into the lower plenum, and steam cooling experiments with droplet injection to simulate dispersed flow film boiling.

The facility housing is constructed of Inconel for dimensional stability at elevated temperatures. There are five pairs of large windows on opposite sides of the housing such that video cameras can be used to film the reflood process as well as using a laser illuminated digital camera system which is used to measure the entrained drop velocity and drop diameters within the rod bundle. The windows are also centered on the axial location of the spacer grids that are used in the bundle to hold and position the electrical heater rods. In this manner, the flow behavior upstream and downstream of the spacer grids can be observed.
The electrical heater rods used in the facility have an outside diameter of 9.5-mm (0.374-inches) and represent a portion of a typical 17 x 17 fuel rod array. The heater rods are single ended such that the top ground plate, which hold the heaters, represents the top nozzle of a fuel assembly.

The axial power shape represents a top skewed power shape. The power during reflood experiments was kept constant to obtain quasi-steady state condition which provides data for a long period of time before quenching.

Figure 5-4 shows the cross-section of the rod bundle. There are 45 heated electrical rods and four unheated Inconel support rods in the corners of the test assembly which are used to support the spacer grids as well as to bring out instrumentation which is located within the bundle. There are approximately 500 channels of instrumentation for the facility.

To measure the sub-channel vapor superheat, two types of miniature thermocouple probes were used. One type of miniature thermocouple probe was suspended from the spacer grids and would face into the flow. There were three miniature thermocouples that were attached to the spacer grid and faced into the flow for the higher elevation grids were vapor superheat would be present. The thermocouples were 0.38 mm in diameter and were supported by 2.44 mm Inconel tubes where were tack welded to the spacer grids. Also, there were 0.38 mm thermocouples which were spot welded to the spacer grids to measure the grid temperature to determine if the grid was hot and dry or wetted and quenched. These thermocouple leads were also routed to the corner hollow tubes and out of the bundle.
The second vapor superheat measurement uses a traversing thermocouple rake consisting of three, 0.38 mm thermocouples which were attached to a thin piece of Inconel shim stock and to a tube (which was out of the flow stream) that could be moved to different radial positions within the bundle. The thermocouples were located one rod pitch apart on the Inconel rake such that they could measure three vapor temperatures at different radial positions within the bundle. The probe could be positioned to measure vapor temperatures in the center of the sub-channel or points in between the sub-channel centers. There are a total of 13 of the traversing thermocouple rakes at different axial positions, most of which can be powered by a stepping motor to traverse the flow. The traversing rakes are located at different axial positions between the spacer grids such that the effects of the spacer grids on the heat transfer and droplet break-up can be determined.
A laser illuminated digital camera system, developed by Penn State and Oxford lasers is being used to measure the entrained drop size and velocity at different axial positions within the bundle in the dispersed flow film boiling regime.

To date only forced reflood tests, radiation only experiments and interfacial drag tests have been performed on the Rod Bundle Heat Transfer Facility. Steam cooling and steam cooling with droplet injection tests will be performed in the near future. Several forced reflood experiments have been performed over a range of inlet flooding rates, pressures, inlet sub-cooling, initial heater rod temperatures, and heater rod peak power levels. Repeat experiments have been performed for the same test conditions such that the laser illuminated camera system could be moved to different axial locations within the bundle to obtain additional data on the entrained droplet sizes. Also, the traversing steam probes were moved to a different radial position for repeat experiments to obtain additional data on the vapor temperature distribution within the rod bundle.

The initial results indicate that the spacer grids can significantly improve the dispersed flow heat transfer downstream of the grids by a combination of increased turbulent mixing as well as shattering of the entrained droplets in the highly non-equilibrium dispersed two-phase flow.

5.2.1 RBHT Run 1383

Table 5-3 summarizes the conditions for RBHT Run 1383, which was conducted at the same pressure as the FLECHT SEASET Run 31504. As stated earlier, like the all the other RBHT reflood tests, run 1383 was conducted at constant power.
Table 5-3: Conditions for RBHT Run 1383

<table>
<thead>
<tr>
<th>Condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Upper Plenum Pressure</td>
<td>0.27 MPa</td>
</tr>
<tr>
<td>Initial Clad Temperature at 2.74m</td>
<td>1040 K</td>
</tr>
<tr>
<td>Rod Peak Power</td>
<td>1.3 kW/m</td>
</tr>
<tr>
<td>Cold Fill Rate</td>
<td>25.4 mm/sec</td>
</tr>
<tr>
<td>Injected Coolant Temperature</td>
<td>393 K</td>
</tr>
</tbody>
</table>

5.2.2 RBHT Run 1096

RBHT Run 1096 was at a lower pressure than RBHT Run 1383 and at a similar flow rate. Table 5-4 summarizes the conditions for RBHT Run 1096.

Table 5-4: Conditions for RBHT Run 1096

<table>
<thead>
<tr>
<th>Condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Upper Plenum Pressure</td>
<td>0.14 MPa</td>
</tr>
<tr>
<td>Initial Clad Temperature at 2.74m</td>
<td>1040 K</td>
</tr>
<tr>
<td>Rod Peak Power</td>
<td>1.3 kW/m</td>
</tr>
<tr>
<td>Cold Fill Rate</td>
<td>25.4 mm/sec</td>
</tr>
<tr>
<td>Injected Coolant Temperature</td>
<td>393 K</td>
</tr>
</tbody>
</table>


5.3 Comparison of FLECHT-SEASET and RBHT Experiments

FLECHT-SEASET and RBHT experiments are both bundle reflood tests but they have several differences which must be taken into account for the analysis, code modeling and code evaluation. These differences can be listed as:

1. The RBHT runs have an axial power shape that is peaked towards the top of the bundle. This power shape is useful to represent a more limiting power shape.
2. The peak power levels in the RBHT series of tests are significantly lower than those set in the FLECHT SEASET series.
3. The FLECHT SEASET tests were conducted with an ANS 1979 decaying power. The RBHT series of tests were performed at constant power. This makes the RBHT test duration longer.
4. The inlet subcooling condition for the FLECHT-SEASET tests are higher for most of the tests.

These differences in the tests and their effects on the analysis of the data are discussed further in Chapter 6. The code evaluation and the results of COBRA-TF analysis for the selected RBHT and FLECHT-SEASET tests are described in Chapter 6.
Chapter 6

EVALUATION OF THE PROPOSED COBRA-TF FIVE-FIELD SOLUTION FOR DISPERSED FLOW FILM BOILING

The proposed models and the new solution method proposed to simulate the dispersed flow film boiling with COBRA-TF code are described in Chapter 4. The modifications proposed in Chapter 4 must be evaluated using proper sets of data obtained for the rod bundle reflood tests. The data from rod bundle experiments are particularly useful to evaluate the proposed models and method since the bundles used contain prototypical spacer grids. The FLECHT-SEASET and RBHT rod bundle reflood tests selected for the evaluations are described in Chapter 5.

In order to present the improvements in code’s prediction capability for dispersed flow film boiling characteristics, comparisons have been performed between the original and the modified COBRA-TF simulations and experimental data. For the code evaluation, measurements and COBRA-TF estimations for the rod temperatures, steam temperatures, grid temperatures, quench front locations for the heater rods and spacer grids are compared. In addition to these comparisons, to demonstrate the need for improving initial drag models used in COBRA-TF, amount of liquid mass stored in the bundle during the experiments is compared with the calculated liquid mass stored.

The comparisons for two experiments performed during RBHT project are presented. The RBHT experiments are longer experiments and the dispersed flow film boiling exists longer in these experiments. The COBRA-TF input model prepared for the simulations of RBHT experiments are also presented.

The results of the comparisons performed for the selected FLECHT-SEASET experiments are presented in a separate section. This section also includes the COBRA-TF input modeling. Since the grid temperatures were not measured during the FLECHT-
SEASET experiments, comparisons for the grid temperatures and the quench locations are not included in the modified COBRA-TF evaluation.

In addition to evaluation analysis, the sensitivity of the code results to change in interfacial drag is presented.

6.1 Analysis of the RBHT Reflood Data

As mentioned earlier, two experiments of the RBHT reflood tests were selected for the code evaluation and assessment of the modifications performed. Since many of the COBRA-TF reflood models are either developed or validated against the data obtained during FLECHT-SEASET tests, evaluating the code modifications using different rod bundle experiment data is very important and interesting in the analysis point of view.

For the analysis, a one-channel bundle modeling is used as an input model for COBRA-TF. This modeling is described in Section 6.1.1 and code evaluation using two RBHT experiments is summarized in sections 6.1.2 and 6.1.3.

6.1.1 One Channel RBHT Bundle Model Input Deck Description

The analysis performed with the one channel model includes modeling of the 7 x 7 rod array having 45 heater rods, 4 unheated rods and housing which surrounds the bundle.

Figure 6-1 presents the schematic for the model. As presented in Figure 6-1, model contains two sections. Section 1 includes one channel and it examines the actual heated length of the bundle 3.66 m (144 in.) and unheated length 0.127 m (5.0 in.). This channel includes all rods.
In order to model the heated rods, unheated rods and housing, three geometry types were employed. The *hrod* type is used for solid cylinders; therefore this type is used to model heated rods. The heater rods are composed of 4 layers. These layers include Boron Nitride, Monel K-500, Boron Nitride and Inconel 600.

The unheated rods were defined by using *tube* geometry. Unheated rods are made of Inconel 600. Housing was modeled as a single wall having a cross sectional area and wetted perimeter equal to the sum of four individual sides. The 0.102 m (4 in.) thick molded perlite-WR1200 insulator was included in the model. Figures 6-2 and 6-3 present the dimensions and materials for the heated rods, unheated rods and housing.

Axially, 32 nodes in total were specified. For the axial nodalization in section 1, refined mesh option was selected. This yields more accurate results.

There are 7 grids in the bundle. The grids are located at 0.108, 0.698, 1.22, 1.74, 2.26, 2.78, 3.305 m (4.25, 27.42, 47.87, 68.5, 88.97, 109.52, 130.11 in.) elevations measured from the bottom of the heated length.

The axial power shape used in these analyses is shown in Figure 6-4. In all the RBHT calculation performed to this date models constant power during the transient.

A sample input deck for the RBHT reflood test Experiment 1383 is presented in Appendix A.
Figure 6-1: Partitioning of Channels and Sections
Figure 6-2: RBHT Heated Rod Materials and Nodalization

Figure 6-3: RBHT Unheated Rod and Housing Nodalization
6.1.2 Analysis of RBHT Experiment 1383

Experiment 1383 of the Rod Bundle Heat Transfer tests is the reference test analyzed for the 0.28 MPa (40 psia) case. The inlet flooding rate for this test is 25.4 mm/sec (1.0 in/sec), peak power is 1.3 kW/m (0.4 kW/ft), initial clad temperature is 760 °C (1400 °F) and inlet subcooling is 11 °C (20 °F). With these properties, 1383 is the longest experiment in 0.28 MPa (40 psia) tests. This experiment is analyzed both with the original and modified COBRA-TF.

Figure 6-5 presents the comparison for the clad temperature measurement, and code’s predictions. The elevation for which the comparisons are presented is the location with

Figure 6-4: RBHT Rod Bundle Axial Power Distribution
the highest temperature and power and it is right below the 6th grid. The measurement error in temperature is also presented on Figure 6-5.

As Figure 6-5 presents, both the original code and the modified code do not agree well with the measurement when it comes to quenching of the given elevation. The agreement between the prediction of the modified COBRA-TF and the experimental measurements is good for the first 180 seconds of the transient. For this time period, as Figure 6-5 indicates code prediction is improved comparing the temperature calculated by the original code. This is due to modeling of small droplets. Since the small droplets are dragged by the vapor quickly out of the bundle, there is less water remained in the bundle. This causes higher temperature prediction for the rod and vapor. Figure 6-6 presents the comparisons for the vapor temperature at 2.54 m (100 in.). As Figure 6-6 indicates, agreement between the vapor temperature measurement and modified code prediction is improved.

As Figure 6-5 presents, After 180 seconds, code starts under predicting the rod temperature. This is also true for the vapor temperature as presented in Figure 6-6. Both figures indicate that the interfacial heat transfer between the vapor and liquid phases is predicted higher. This indicates that the amount of liquid in the bundle is predicted higher.
Figure 6-5: Heater Rod Temperature at 2.69 m (106.0 in.), Experiment 1383
Comparisons for the heater rod and vapor temperatures at 1.9 m (74.9 in.) location are presented in Figures 6-7 and 6-8. As Figure 6-7 indicates, the agreement between the measurement and the code predictions is improved with the modifications performed in this study. An earlier quench is predicted by the code for this location although the vapor temperature is predicted higher as Figure 6-8 indicates. This disagreement between the data and the code predictions presents the need to improve the models used for boiling curve, especially for the minimum film boiling temperature model. As Figure 6-7 indicates, both the original and modified codes predict higher minimum film boiling temperature than the experimental value, this causes earlier quenching of the rods.

Figure 6-6: Steam Temperature at 2.54 m (100.0 in.), Experiment 1383
Figure 6-7: Heater Rod Temperature at 1.9 m (74.9 in.), Experiment 1383
Figure 6-8: Steam Temperature at 1.9 m (74.9 in.), Experiment 1383
Figure 6-9 presents the results of the heater rod temperature comparisons for the 2.79 m (110 in.) location. This location is right above the 6th spacer grid of the bundle, and therefore the comparisons performed for this elevation is useful to present the effects of spacer grids on the heat transfer. The heater rod temperature at this elevation is predicted higher by both the original code and the modified code. Figure 6-10 presents the heater rod temperatures measured at 2.69 m and 2.79 m (106.0 in. and 110.0 in.) locations, which are located upstream and downstream of the 6th spacer grid. This figure indicates the temperature drop between these two elevations is about 100 °C (170 °F). The modified code predictions for the rod temperatures at the 2.69 m and 2.79 m (106.0 in. and 110.0 in.) locations are presented in Figure 6-11. Code predicts only about 50 °C (100 °F) drop for the upstream and downstream of the grid 6.

Figure 6-9: Heater Rod Temperature at 2.79 m (110.0 in.), Experiment 1383
Figure 6-10: Heater Rod Temperatures above and below the 6th Spacer Grid - Data, Experiment 1383
Comparison between Figures 6-10 and 6-11 indicates that the spacer grid effects on the heat transfer must be analyzed. Figure 6-12 presents the temperature measurement and original and modified code predictions for the temperature of the 6th spacer grid. As the figure presents, although it is improved, a late grid quench and higher grid temperature is predicted by the modified code. When Figures 6-10 and 6-12 are examined together, it can be presented that the temperature heat-up at 2.79 m (110.0 in.) location ends when spacer grid temperature starts decreasing then grid is finally quenched.

The code predictions presented in Figures 6-11 and 6-12 show the same behavior; both spacer grid and heater rod heat up at the beginning of the transient, then when spacer grid temperature starts decreasing, heater rod temperature heat-up ends. Once the grid is predicted to be quenched, the heater rod temperature at 2.79 m (110.0 in.) elevation starts decreasing.

Figure 6-11: Heater Rod Temperatures above and below the 6th Spacer Grid – Modified Code, Experiment 1383
In order to obtain a better agreement between the code predictions and experimental data for the drop in heater rod temperature due to spacer grid, a better prediction of grid quench is required. Although the modifications in this study performed for the spacer grid models lead to improvement, code predictions can be further improved by adding a minimum film boiling temperature model for the spacer grids and by modifying the heat transfer coefficients used to limit the quench velocity. The heat transfer coefficients used for the spacer grids are described in Chapter 4.

Figure 6-12: Grid Temperature, Grid 6, Experiment 1383

![Temperature, Grid 6, Experiment 1383](image)

Figure 6-13 presents the quench front location calculated by the original and modified codes and the quench front location determined using experimental data. For the experiment, quench front location is determined using a simple FORTRAN code.
This code and calculations performed to determine the quench location are described in Appendix B.

Figure 6-13 indicates that by adding a new droplet field, code predictions for the quench front location is improved. The small droplets are quickly dragged out of the bundle by the vapor and this helps decreasing the liquid mass stored in the bundle, therefore a better agreement for the quench location is obtained by the modified code.

The comparison of the measured and calculated liquid mass stored in the bundle is presented in Figure 6-14. As Figure 6-14 indicates, the calculated liquid mass in the bundle is much higher than the liquid mass measured. Once too much liquid is predicted in the bundle, a lower steam temperature, therefore lower heater rod temperature is calculated and quench front progress becomes fast.

Figure 6-14 presents the need for improvement in interfacial drag models for the droplet fields. As Figure 6-14 indicates, interfacial drag calculated by the code is small.
Figure 6-13: Quench Front Location, Experiment 1383
6.1.3 Analysis of RBHT Experiment 1096

The reference test analyzed for the 0.14 MPa (20 psia) case for the Rod Bundle Heat Transfer tests is experiment 1096. The inlet flooding rate for this test is 25.4 mm/sec (1.0 in/sec), peak power is 1.3 kW/m (0.4 kW/ft), initial clad temperature is 760 °C (1400 °F) and inlet subcooling is 11 °C (20 °F). Since dispersed flow film boiling is observed for a longer period of time during 1096, the analysis of this experiment both with the original and modified COBRA-TF is important.

The comparisons of the clad temperature, vapor temperature and grid temperature present that with the modified code, code’s estimations for rod bundle reflood are improved.

Figure 6-14: Liquid Mass Stored in the Bundle, Experiment 1383
The comparison for the clad temperature measurement and code’s temperature predictions for the location with the highest temperature and power is presented in Figure 6-15. The selected location is right below the 6\textsuperscript{th} grid.

As Figure 6-15 presents, the agreement between the code simulations and the data for quenching of the given elevation is improved with the modified code. For the first 250 seconds of the transient the agreement between the prediction of the modified COBRA-TF and the experimental data is good. As explained in section 6.1.2, the improvement in the code’s predictions is due to modeling of small droplets. Since the small droplets are carried out of the bundle by the vapor, there is less water remained in the bundle.

Figure 6-16 presents the comparisons for the vapor temperature at 2.54 m (100.0 in.) location. As Figure 6-16 indicates, agreement between the vapor temperature measurement and modified code prediction is improved. Figures 6-15 and 6-16 present that both rod temperature and vapor temperature is under predicted.
Figure 6-15: Heater Rod Temperature at 2.69 m (106.0 in.), Experiment 1096
Comparisons for the heater rod and vapor temperatures at 1.9 m (74.9 in) location are presented in Figures 6-17 and 6-18. As Figure 6-17 indicates, the agreement between the measurement and the code predictions is improved with the modifications performed in this study. The early quench for this elevation is predicted by the code. This presents the need to improve the models used for boiling curve, especially for the minimum film boiling temperature model.

Figure 6-16: Steam Temperature at 2.54 m (100.0 in.), Experiment 1096

Comparisons for the heater rod and vapor temperatures at 1.9 m (74.9 in) location are presented in Figures 6-17 and 6-18. As Figure 6-17 indicates, the agreement between the measurement and the code predictions is improved with the modifications performed in this study. The early quench for this elevation is predicted by the code. This presents the need to improve the models used for boiling curve, especially for the minimum film boiling temperature model.
Figure 6-17: Heater Rod Temperature at 1.9 m (74.9 in.), Experiment 1096
Figure 6-19 presents the results of the heater rod temperature comparisons for the location right above the 6\textsuperscript{th} spacer grid of the bundle. The heater rod temperature at this elevation is predicted higher by both the original code and the modified code. In order to present the effect of the spacer grid, Figure 6-20 presents the heater rod temperatures measured at 2.69 m and 2.79 m (106.0 in. and 110.0 in.) in locations, which are located upstream and downstream of the 6\textsuperscript{th} spacer grid. This figure indicates the temperature drop between these two elevations is about 130 °C (240 °F). The modified code predictions for the rod temperatures at the 2.69 m and 2.79 m (106.0 in. and 110.0 in.) locations are presented in Figure 6-21. Code predicts only about 70 °C (125 °F) drop for the upstream and downstream of the grid 6.
Figure 6-22 presents the temperature measurement and original and modified code predictions for the temperature of the 6th spacer grid. As the figure presents, although it is improved, a late grid quench and higher grid temperature is predicted by the modified code. When Figures 6-20 and 6-22 are examined together, it can be presented that the temperature heat-up at 2.79 m (110.0 in.) location ends when spacer grid temperature starts decreasing. Once grid is finally quenched, a drop in the heater rod temperature is measured.

The code predictions presented in Figures 6-21 and 6-22 show the same behavior; both spacer grid and heater rod heat up at the beginning of the transient, then when spacer grid temperature starts decreasing, heater rod temperature heat-up ends. Once the grid is predicted to be quenched, the heater rod temperature at 2.79 m (110.0 in.) elevation starts decreasing.
Figure 6-19: Heater Rod Temperature at 2.79 m (110.0 in.), Experiment 1096
Figure 6-20: Heater Rod Temperatures above and below the 6th Spacer Grid – Data, Experiment 1096
Figure 6-21: Heater Rod Temperatures above and below the 6th Spacer Grid – Modified Code, Experiment 1096
Figure 6-23 presents the quench front location calculated by the original and modified codes and the quench front location determined using experimental data. Figure 6-23 indicates that by adding a new droplet field, code predictions for the quench front location is improved.

The comparison of the measured and calculated liquid mass stored in the bundle is presented in Figure 6-24. As Figure 6-24 indicates, the calculated liquid mass in the bundle is much higher than the liquid mass measured. Figure 6-24 also presents the need for improvement in interfacial drag models for the droplet fields.
Figure 6-23: Quench Front Location, Experiment 1096
6.2 Analysis of the FLECHT-SEASET Reflood Data

As mentioned in Chapter 5, two experiments of the FLECHT-SEASET reflood tests were selected for the code evaluation and assessment of the modifications performed. Many of the COBRA-TF reflood models are either developed or validated against the data obtained during FLECHT-SEASET tests.

For the analysis, a one-channel bundle modeling is used as an input model for COBRA-TF. This modeling is described in Section 6.2.1 and code evaluation using two FLECHT-SEASET experiments is summarized in sections 6.2.2 and 6.2.3. Since the grid
temperatures were not measured, for the code evaluation using FLECHT-SEASET data, grid temperature measurements are not presented here.

**6.2.1 One Channel FLECHT-SEASET Bundle Model Input Deck Description**

The analysis performed with one channel model includes the modeling of 161 heater rods, 16 unheated rods and housing which surrounds the bundle. The model contains 2 sections.

Section 1 includes one channel and it examines the actual heated length of the bundle. This channel includes all rods.

In order to model the heated rods, unheated rods and housing, three geometry types were employed. The *hrod* type is used for solid cylinders, therefore this type is used to model heated rods.

Heater rods are composed of 4 material layers. These layers include Boron Nitride, Monel K-500, Boron Nitride and Stainless Steel 316.

The unheated rods were defined by using *tube* geometry. Unheated rods are made of Stainless Steel 316.

Housing was modeled as a single wall having a cross sectional area and wetted perimeter equal to the sum of four individual sides. Housing is made of Stainless Steel 316.

There are 18 radial nodes in heater rods. Axially, 28 nodes in total were specified. For the axial nodalization in section 1, refined mesh option yielding more accurate results was selected. There are 8 grids in the bundle.
6.2.2 Analysis of FLECHT-SEASET Experiment 31504

FLECHT-SEASET experiment 31504 and RBHT experiment 1383 are similar when the pressure and inlet flow rate conditions are compared. Experiment 31504 is a 0.28 MPa (40 psia) experiment with an inlet flow rate of 24.0 mm/sec (0.97 in/sec). The initial peak rod temperature is 863 °C (1585°F). The peak power for 31504 is 2.3 kW/m (0.7 kW/ft) and. The inlet subcooling is 79 °C (144 °F).

The comparison for the clad temperature measurement and code’s temperature predictions for the 2.28 m (90.0 in.) location is presented in Figure 6-25. The selected location is at the upper half of the rod, therefore above the peak power location. As Figure 6-25 indicates, the first 170 seconds of the simulation, the modified code predictions and the data agree well. The original code has better agreement with the data for the quenching of the location. The original code under predicts the heater rod temperature through the entire simulation.

Figure 6-26 presents the comparison for the heater rod predictions and data for 2.13 m (84.0 in.) location. The figure indicates that the original code under predicted the rod temperature, however, the agreement between the data and the original code for the rod quenching is good. Since 31504 is a short experiment, for the simulation of this experiment, errors in model sets do not amplify through the simulation time. Since the modified code models the small droplets that are carried by the vapor out of the bundle, it over predicts the quenching time for this location.
Figure 6-25: Heater Rod Temperature at 2.28 m (90.0 in.), Experiment 31504
The vapor temperatures at higher elevations, at 2.44 m (96.0 in.) and 3.05 m (120.0 in.) are presented in Figures 6-27 and 6-28. These figures indicate that due to small droplets carried out of the bundle, steam temperature is over predicted for both locations by the modified code.

Comparison for the quench front location is presented in Figure 6-29 and it indicates that the bundle quench front progress is over predicted by the modified code. Since adding small droplet field decreases the amount of liquid in the bundle and therefore delays the bundle quench, bundle quench time estimated by the modified code is greater than the measured quench time.

Figure 6-26: Heater Rod Temperature at 2.13 m (84.0 in.), Experiment 31504
Figure 6-27: Steam Temperature at 2.44 m (96.0 in.), Experiment 31504
Figure 6-28: Steam Temperature at 3.05 m (120.0 in.), Experiment 31504
6.2.3 Analysis of FLECHT-SEASET Experiment 31922

Experiment 31922 is a 0.14 MPa (20 psia) experiment with initial peak rod temperature of 883 °C (1621°F). FLECHT-SEASET experiment 31922 and RBHT experiment 1096 are similar when the inlet flow rate and rod peak power conditions are compared. The peak power for 31922 is 1.3 kW/m (0.4 kW/ft) and inlet flow rate is 27.2 mm/sec (1.07 in/sec). The coolant temperature is 35 °C (95 °F), therefore the inlet subcooling is 74 °C (133 °F)

Figure 6-29: Quench Front Location, Experiment 31504
The comparison for the clad temperature measurement and code’s temperature predictions for the 2.28 m (90 in.) location is presented in Figure 6-30. The selected location is at the upper half of the rod, therefore above the peak power location.

Figure 6-30 shows that the first 220 seconds of the transient, agreement between the code predictions and the data is improved with the modified code. The original code has better agreement with the data for the quenching of the location, however it under predicts the rod temperature.

The modified code predicts a longer time for the bundle quench. Since the flow inlet subcooling is high and axial power shape is cosine for experiment 31922, it is a short experiment and bundle quenches quickly. In this case, dispersed flow film boiling does not exist for a long period of time and since adding small droplet field decreases the amount of liquid in the bundle and therefore delays the bundle quench, bundle quench time estimated by the modified code is greater than the measured quench time.

The comparison for the heater rod predictions and data for 2.13 m (84.0 in.) is presented in Figure 6-31. The figure indicates that the temperature heat up is followed by the fast decrease in rod temperature although the location selected is close to axial peak power location. For this location, the original code under predicted the rod temperature but its prediction for quench time agree with the data, too. Modified code, which models the small droplets that are carried by the vapor out of the bundle, over predicts the quenching time for this location.
Figure 6-30: Heater Rod Temperature at 2.28 m (90.0 in.), Experiment 31922
Figures 6-32 and 6-33 present the comparisons for the vapor temperatures at high elevations, at 2.44 m and 3.05 m (96.0 in. and 120.0 in.) These figures indicate that steam temperature is over predicted for both locations by the modified code. This is again due to droplets carried out of the bundle.

Comparison for the quench front location is presented in Figure 6-34 and it indicates that for the upper half of the bundle quench front progress is over predicted by the modified code.
Figure 6-32: Steam Temperature at 2.44 m (96.0 in.), Experiment 31922
Figure 6-33: Steam Temperature at 3.05 m (120.0 in.), Experiment 31922
6.3 Simulation of the RBHT Experiment 1096 with Elevated Interfacial Drag Coefficient

The analysis performed for the RBHT experiments with long duration present that the interfacial drag model for the drop and liquid phases must be modified. In order to determine the sensitivity of results to the interfacial drag models, the interfacial drag coefficient for the large and small drop fields were increased and RBHT experiment 1096 was simulated.

Figure 6-35 presents the heater rod temperature measured and calculated for the peak power location. In Figure 6-35, measured rod temperature, the rod temperature calculated with the modified code and temperature calculated with the modified code
with increased drag coefficient are presented. As Figure 6-35 indicates, increasing interfacial drag results in an increase in the quench time for the location. Once the interfacial drag increases, the amount of liquid carried out of the bundle increases and since there will be less liquid in the bundle, quench font progression gets slower. This would also cause a decrease in interfacial heat transfer from drop fields to vapor field and therefore causes a decrease in heat transfer coefficient for the continuous vapor.

Figure 6-35: Heater Rod Temperature at 2.69 m (106.0 in.), Experiment 1096, Comparison of Data, Modified Code Calculation and Modified Code Calculation with Altered Interfacial Drag Coefficient
Best estimate computer codes are useful tools for the reactor accident analysis and therefore the development, modification and upgrade of these codes are very important. To increase the predictive capability of the codes such as COBRA-TF in simulating postulated loss of coolant accident analysis, new models or modifications are required. The summary of such models and experiment for modeling reflood phase of the loss of coolant accident, especially the dispersed flow film boiling heat transfer mode is described in Chapter 2.

Since this study was performed to modify COBRA-TF by adding a new droplet field, a brief description of COBRA-TF equations, derivation of these equations and closure models are summarized in Chapter 3. This chapter also aims to provide a comprehensive documentation for COBRA-TF code to help future users.

In Chapter 4, the modifications and models proposed in this study are described. The equations added and modified and the new models to be implemented for the new field are discussed in this chapter. The presence of small droplets in the dispersed flow causes a decrease in the liquid amount stored in the bundle or in a real accident case in the core since the smaller droplets are easily dragged by the vapor. The mass, energy and momentum exchange between small droplet field and the other fields, and models for these exchanges used in this study are summarized in Chapter 4. The entrainment, large drop break up, interfacial drag and heat transfer models are described in detail in this chapter.

In addition to modifications performed to add a new field, the changes to spacer grid models to improve code’s predictions for spacer grid effects on heat transfer are described in Chapter 4.

In Chapter 5, the reflood experiments selected to evaluate the code modifications are described. In order to perform analysis with the modified code, two reflood
experiment sets performed for rod the bundles, FLECHT-SEASET and RBHT tests are selected. In Chapter 5, the geometries of the FLECHT-SEASET and RBHT bundles and test conditions for the selected experiments are summarized. In total 4 experiments were selected for the code evaluation study. These experiments are reference reflood tests with low pressure, low inlet flooding rate conditions.

The results of the code simulations for the selected experiments are discussed in Chapter 6. The results presented in Chapter 6 indicates that adding small droplet field to COBRA-TF code improves code predictions for long reflood transients such as RBHT experiments since during these tests dispersed droplet film boiling heat transfer mode exists longer. For the RBHT experiments, the improvements in heater rod temperature, vapor temperature and quench front progression calculations are presented. The results of the analysis performed for the RBHT experiments presented that, calculations performed with the modified code for the spacer grid temperature and grid quench are in good agreement with the data.

The work presented in this dissertation had the goal of improving the dispersed flow film boiling model in COBRA-TF. This was achieved by the novel concept of adding small droplet field, which is generated by the large droplet break up due to spacer grids and by forces acting on large drops, as well as the entrainment. The purpose of adding this new field was to model interfacial drag and heat transfer between the vapor field and droplet fields more accurately. In addition to new continuity equations, this is achieved by adding a new interfacial area transport equation for the small droplet field. The comparisons performed with the RBHT reflood experiments and code predictions presented that the goal set for this work has been accomplished. The results of this work also presented the new research needed for the further improvement of the code.

One of the most important needs for the future research is the improvement of the interfacial drag models not only for the drop fields but also for the continuous liquid film. The results of the non-physical code alteration for the interfacial drag coefficient performed in this study presented this need. With better interfacial drag models, the velocity of both vapor and liquid fields would be modeled accurately. Since the mass, momentum and energy transfer between the vapor field and drop fields and drop velocity
and size distribution depend on the interfacial drag between these fields, an accurate model is needed. A new interfacial drag model can be easily implemented in COBRA-TF since it already has the interfacial area transport equations for the drop fields.

The calculations performed for the liquid mass stored in the bundle indicates that, besides the interfacial drag model, the entrainment model, especially the entrainment at the quench front must be revised. A lagrangian solution for the quench front, better modeling of mass transfer and momentum below quench front and detailed analysis of bundle reflood tests to determine liquid transfer would be helpful further research to improve the entrainment model.

Another future research is needed for the heat transfer package of the code. The results of this study present that especially the minimum film boiling temperature model of the code must be revised and modified. Once the minimum film boiling temperature is determined, it is used to determine the heat transfer mode. Code uses the transition boiling heat transfer package when the rod temperature becomes smaller than the minimum film boiling temperature. Therefore the switch from film boiling to transition boiling is determined by the minimum film boiling temperature.

The need for the improvement in heat transfer package has been also presented by the spacer grid analysis. The minimum film boiling temperature set for the spacer grids and the heat transfer coefficients used to determine the quench velocity must be revised and modified. The spacer grid rewet affects both the vapor temperature and rod temperature, therefore accurate modeling of spacer grid rewet is important for the accurate modeling of dispersed droplet film boiling heat transfer mode.

The reflood package of the COBRA-TF code must be revised and tested. There are several numerical values assigned to interfacial heat transfer and interfacial drag coefficients to set the minimum and maximum values. These numerical values are based on experiments and the sensitivity of the code solutions and stability to these values must be determined.

As mentioned earlier, in order to add the new droplet field, the mass equation for this field was added to the code’s solution scheme by increasing the size of the Jacobian matrix. An accurate and computationally less expensive methodology must be developed.
and implemented to the code. If needed, such methodology would make adding new fields to the code easier.


[15]. FEBA Data Reports 1 & 2, (March 1984).


[23]. Data has not been published yet.


[33]. Yoder, G. L., “Heat Transfer Near Spacer Grids in Rod Bundles”, Oak Ridge Natl Lab, Oak Ridge Tenn, USA.

[34]. Iguchi, T., Ohzimu, S., Anoda, Y., “Grid Spacer Effect on Reflood Behavior Observed at Reflood Experiment with 5x5 Bundle Test Section Under Wide Pressure Condition”, Submitted to 7th International Conference on Nuclear Engineering, Tokyo, Japan, (1999).


[36]. Yoder, G. L., “Heat Transfer Near Spacer Grids in Rod Bundles”, Oak Ridge Natl Lab, Oak Ridge Tenn, USA.


Appendix A

Sample COBRA-TF Input Deck

*******************************************************************************
********
* INPUT DECK
* RBHT Single Channel Model
* Reflood 1 in/sec
* No Radiation H.T.
* this input models the rbht experiment 40 psia
*
* S. ERGUN june 2002
*******************************************************************************
********

*************************************** ********************************
********
* Main Problem Control Data             *
*******************************************************************************
********

** ICobra
  0
* DSTEP   TIMET
   0    0.0
* EPSO   OITMAX   IITMAX
   .001   10      40
* INIT             TEXT
     1
**** Reflood 1 in/sec Bundle Rod 7x7
****
*
*******************************************************************************
********
* Group 1 - Calculation Variables and Initial Conditions
*
*******************************************************************************
********
*NGRP NGAS  
   1   1
* PREF   HIN   GIN   AFLUX   GHIN   VFRAC1   VFRAC2
RSBF
  40.0  1170.  0.0  0.270   124.  0.0   .9999
1.0
*GTYPE   VFRAC
air    .0001
     1
*******************************************************************************
********
* Group 2 - Channel Description
*
*NGRP NCHA
2 2
* I AN PW ABOT ATOP NMGP NRRD DROD
17.21971.74 0.0 0.0 45.374
250.2471.74 7.22 0.
*

Group 3 - Transverse Channel Connection (Gap) Data
*

*NGRP NK
3 0
* K IK JK GAPN LNGT WKR FWAL IGPB IGPA FCT IGAP JGAP
1 1 21.952 1.38 2.0 0.0 0 0 1.0 0 0
*GMLT ETNR
* 16.0.0
*NMGP
0
*

Group 4 - Vertical Channel Connection Data
*

*NGRP NSEC NSIM IREB
4 2 1 0
*ISEC NCHN NONO DXS IVAR
1 1 33 4.653 33
*JLEV VARDX JLEV VARDX JLEV VARDX VARDX
2 5.0 3 4.25 4 4.375 5 4.375 6
3.605 7 3.605 8 3.605 9 3.605 10 4.79 11
4.79 12 6.0 13 4.87 14 5.13 15 4.0 16
6.0 17 5.0 18 4.0 19 4.0 20 4.0 21
5.0 22 4.0 23 5.5 24 5.0 25 5.0 26
3.0 27 4.0 28 3.0 29 4.0 30 6.0 31
4.0 32 4.8 33 4.6 34 6.0
* I KCHA KCHB
1 2 1
* 2 3 2
*ISEC NCHN NONO DXS IVAR
2 1 3 4.0
* I KCHA KCHB
2 2 1
*IWDE
* Group 7 - Local loss coefficient and Grid spacer data

<table>
<thead>
<tr>
<th>NGRP</th>
<th>NCD</th>
<th>NGT</th>
<th>IFGQ</th>
<th>IFSD</th>
<th>IFES</th>
<th>IFTP</th>
<th>NFBS</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>7</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>0</td>
</tr>
</tbody>
</table>

* nglcof

<table>
<thead>
<tr>
<th>CDL</th>
<th>J</th>
<th>ICDM</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.2</td>
<td>3</td>
<td>1</td>
</tr>
<tr>
<td>1.2</td>
<td>9</td>
<td>2</td>
</tr>
<tr>
<td>1.2</td>
<td>13</td>
<td>1</td>
</tr>
<tr>
<td>1.2</td>
<td>17</td>
<td>1</td>
</tr>
<tr>
<td>1.2</td>
<td>22</td>
<td>1</td>
</tr>
<tr>
<td>1.2</td>
<td>26</td>
<td>1</td>
</tr>
<tr>
<td>1.2</td>
<td>31</td>
<td>1</td>
</tr>
</tbody>
</table>

* ING NGAL NGCL IGMT GLOSS GABLOC GLONG GPERIM

<table>
<thead>
<tr>
<th>NNGL</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
</tr>
<tr>
<td>9</td>
</tr>
<tr>
<td>13</td>
</tr>
<tr>
<td>17</td>
</tr>
<tr>
<td>22</td>
</tr>
<tr>
<td>26</td>
</tr>
<tr>
<td>31</td>
</tr>
</tbody>
</table>

500. 803.1025.1135.1228.1239. 945.

* Group 8 - Rod and Unheated Conductor Data

<table>
<thead>
<tr>
<th>NGRP</th>
<th>NRRD</th>
<th>NSRD</th>
<th>NC</th>
<th>NRTB</th>
<th>NRAD</th>
<th>NLTY</th>
<th>NSTA</th>
<th>NXF</th>
<th>NCAN</th>
<th>RADF</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>2</td>
<td>1</td>
<td>2</td>
<td>2</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td></td>
</tr>
</tbody>
</table>

* N IFTY IAXP NRND DAXMIN RMULT RADIAL HGAP ISEC HTMB TAMB

| 1    | 1    | 1    | 2   | 0.05 | 45.  | 1.0  | 5000. | 1   | 0.  |
| 2    | 1    | 1    | 2   | 0.05 | 29.0 | 1.0  | 5000. | 1   | 0.  |
| 2    | 1    | 0    | 2   | 0.05 | 4.   | 0.0  | 5000. | 1   | 0.  |

75.

* NSCH PIE

| 1    | 1.0 |
| 2    | 1   | 1   | 2   | 0.05 | 29.0 | 1.0  | 5000. | 1   | 0   |
| 2    | 1   | 0   | 2   | 0.05 | 4.   | 0.0  | 5000. | 1   | 0   |

75.

* NSCH PIE

| 1    | 1.0 |
| 2    | 1   | 1   | 2   | 0.05 | 29.0 | 1.0  | 5000. | 1   | 0   |

75.

* NSCH PIE

| 1    | 1.0 |
| 2    | 1   | 1   | 2   | 0.05 | 29.0 | 1.0  | 5000. | 1   | 0   |

75.

* NSCH PIE

| 1    | 1.0 |
| 2    | 1   | 1   | 2   | 0.05 | 29.0 | 1.0  | 5000. | 1   | 0   |

75.
1 3 14.20 14.20 1.0 1 0 0.0 75.0
* I NRT1 NST1 NRX1
1 2 0 10
*IRTB
1 2
* AXIALT TRINIT AXIALT TRINIT AXIALT TRINIT AXIALT TRINIT
TRINIT
0. 600.0 13.0 600. 55.00 1139. 65.00
1204.0
105.10 1399.0 111.00 1383.0 133.0 1069. 147.0
766.0
151.01 696.0 156.00 696.0
* I NRT1 NST1 NRX1
2 0 1 6
*ISTB
1
* AXIALT TRINIT AXIALT TRINIT AXIALT TRINIT AXIALT TRINIT
TRINIT
0. 220.0 66.9 555. 113.0 725.0 133.0 549.0
149.1 289.0 155.0 220.0
*
***********************************************************************
***********
* Group 9 - Conductor Geometry Description
*
***********************************************************************
***********
*NGRP NFUE
9 3
* I FTYP DROD DIN NFUE IMOX IMTX
1 hrod .374 0.0 4 0
*NDER MATR TREG QREG NDER MATR TREG QREG NDER MATR TREG QREG
3 2.0685 0.0 3 3 .044 1.0 3 2.0545 0.0 3 1 .02 0.0
* I FTYP DROD DIN NFUE IMOX IMTX
2 tube .374 .209 1 0
*NDER MATR TREG QREG
3 1.0825 0.0
* I FTYP DROD DIN NFUE IMOX IMTX
3 wall 14.20 4.250 2 4 1
*NDER MATR TREG QREG
5 1 .250 0.0 8 4 4. 0.0
*
***********************************************************************
***********
* Group 10 - Axial Power Tables and Forcing Functions
*
***********************************************************************
***********
*NGRP NMAT
10 4
* N NTDP RCOLD IMATAN
1 17 516.7
<table>
<thead>
<tr>
<th></th>
<th>TPROP</th>
<th>CPF1</th>
<th>THCF</th>
<th>TPROP</th>
<th>CPF1</th>
<th>THCF</th>
</tr>
</thead>
<tbody>
<tr>
<td>32</td>
<td>.1010</td>
<td>8.1400</td>
<td>70</td>
<td>.1030</td>
<td>8.340</td>
<td></td>
</tr>
<tr>
<td>212</td>
<td>.1110</td>
<td>9.090</td>
<td>392</td>
<td>.1210</td>
<td>10.04</td>
<td></td>
</tr>
<tr>
<td>500</td>
<td>.1270</td>
<td>11.93</td>
<td>932</td>
<td>.151</td>
<td>15.72</td>
<td></td>
</tr>
<tr>
<td>752</td>
<td>.1410</td>
<td>14.77</td>
<td>1652</td>
<td>.178</td>
<td>14.77</td>
<td></td>
</tr>
<tr>
<td>1000</td>
<td>.158</td>
<td>16.08</td>
<td>1832</td>
<td>.166</td>
<td>16.66</td>
<td></td>
</tr>
<tr>
<td>1292</td>
<td>.193</td>
<td>18.33</td>
<td>1472</td>
<td>.191</td>
<td>15.72</td>
<td></td>
</tr>
<tr>
<td>1500</td>
<td>.193</td>
<td>18.33</td>
<td>1652</td>
<td>.191</td>
<td>15.72</td>
<td></td>
</tr>
<tr>
<td>1832</td>
<td>.216</td>
<td>19.61</td>
<td>2012</td>
<td>.229</td>
<td>18.56</td>
<td></td>
</tr>
<tr>
<td>2192</td>
<td>.241</td>
<td>20.95</td>
<td>2192</td>
<td>.241</td>
<td>19.50</td>
<td></td>
</tr>
</tbody>
</table>

* TPROP CPF1 THCF TPROP CPF1 THCF

212 .1110 9.090 392 .1210 10.04
500 .1270 11.93 932 .151 15.72
752 .1410 14.77 1652 .178 14.77
1000 .158 16.08 1832 .166 16.66
1292 .193 18.33 1472 .191 15.72
1500 .193 18.33 1652 .191 15.72
1832 .216 19.61 2012 .229 18.56
2192 .241 20.95 2192 .241 19.50

2 14 119.
212 .16587 67.37 392 .22014 63.82
572 .26263 60.28 752 .34233 49.64
932 .32194 53.19 1112 .38822 35.46
1292 .39421 50.55 1472 .39891 28.37
1652 .40259 48.33 2012 .40546 21.28
2372 .40822 35.46 2192 .40981 28.37
3 10 528.8
70 .10000 10.083 200 .107 11.33
400 .11400 13.00 600 .117 14.83
800 .120 16.50 1000 .125 18.33
1200 .132 20.00 1400 .141 21.83
1600 .157 23.50 1800 .186 25.16
4 2 12.0

* TPROP CPF1 THCF TPROP CPF1 THCF
650.0 0.2 .058 800. 0.2 .077

*******************************************************
** Group 11 - Axial Power Tables and Forcing Functions
**
*******************************************************
*NGRP NAXP NQ NGPF
11 1 0
*L NAXN
1 6
* Y AXIAL Y AXIAL Y AXIAL
AXIAL
0. 0.0 4.99 0. 5. 0.5 113.00
1.50
149.00 0.5 149.01 0.0
* 0.0 1. 17.5 .921 35.0 .8704 52.5
.8326
* 70. .803 87.5 .7755 105.0 .7512 122.5
.7302
* 140. .714 157.5 .6973 175.0 .6837 192.5
.6710
* 220. .652 255.0 .6332 290.0 .6167 325.0
.6017
* 360. .588 395.0 .5769 430.0 .5656 465.0
.5562
* 500. .547 535.0 .5444 570.0 .5304 605.0
.5243
*     1000.  .002
***********************************************************************
************
* Group 13 - Boundary Conditions Data *
**********************************************************************
************
*NGRP NBND NKBD NFUN NGBD
  13    2    0    0    0    0
*IBD1 IBD2 ISPC NPFN NHFN PVALUE HVALUE XVALUE
  1  1  2  0  0  0.245  215.66  40.0
*HMGA GVAL
  124.  1.0.9999.0001
*NHFN NGFN
  0  0
*IBD1 IBD2 ISPC NPFN NHFN PVALUE HVALUE XVALUE
  2  5  1  0  0  40.0  1170.0  40.0
*HMGA GVAL
  124.  1.0.9999.0001
*NHFN NGFN
  0  0
***********************************************************************
************
* Group 14 - Output Options *
***********************************************************************
************
*NGRP   N1 NOU1 NOU2 NOU3 NOU4 IPRP IOPT IRWR
  14    5    0    0    0    0    1    2    1
0
0
*MAXDP
  9000
*     .0000000001  .01  600.0   1.0  99999999.
*     10.    1.    5.
*     DTMIN (if negative stop)
 - .001
Appendix B

Quench Front Program

In order to determine the quench front progress for the experiments a simple FORTRAN program was developed. As the result of the program, location of the quench front vs time curves were obtained.

The quench front program uses the temperature data measured by rod thermocouples, locations of these thermocouples, and the test time as input. The reflood start time is the starting time for the program. The program requires input data preparation for thermocouple measurements.

A simple criterion is used to determine the quench front location by using the temperature data. One sample of plot for change in rod temperature with time is presented in Figure B-1 and quench time is indicated on the figure.

The first attempt to determine the quench front time was to use a simple numerical approach to get the first derivative for the above data. The time at which the first derivative reaches its maximum value was assumed as the quench time since the rate of change of the heater rod temperature would be the greatest.

To evaluate the first derivative below relation was used:

\[
\frac{dT}{dt} = \frac{T_{n+1} - 2T_i + T_{i-1}}{\Delta t}
\]  

(B-1)

where in the program, \( t \) is 1.0 second.

Figure B-2 presents the change in first derivative of temperature curve with time with two different scales. The figure shows that the first derivative reaches its maximum value approximately 10 seconds after rod quenches. In order to make program to obtain a
more accurate value of the quench time, an additional requirement was added to the program. This requirement determines the quench time when the first derivative of temperature vs time curve is greater than 20.0. The value of 20 was determined by examining most of the rod temperature data for all tests and several elevations.

For the case illustrated in Figure B-2, the quench time was determined as 892 seconds, and the quench temperature for this case is 345 °C (653 ° F) as presented in Figure B-1.

Figures B-3 and B-4 present samples of calculations performed to analyze quench front progress.

As the result of the program, the rod surface temperature at the quenched elevation is also obtained from the data. Figures B-5 and B-6 present the data for quenched temperature versus elevation for experiments 1096 and 1383 respectively. As figures present, the quench temperature data shows a distribution. The mean quench temperatures and standard deviation for these mean temperatures are also presented in the figures. The mean quench temperature for Experiment 1383 is 425.2 °C (788.9 °F) and for Experiment 1096 is 393.75 °C (740.75 °F).
Figure B-1. Sample Clad Temperature Measurement, Experiment 1096

P = 137.9 kPa (20 psia)
Inlet Flooding = 0.0254 m/s (1.0 in/s)
Peak Power = 1.31 kW/m (0.4 kW/ft)
Initial Temperature = 760.0 C (1400 F)
Inlet Subcooling = 11.1 C (20 F)
Figure B-2. Temperature and First Derivative of Temperature Curve, Experiment 1096

P = 137.9 kPa (20 psia)
Inlet Flooding = 0.0254 m/s (1.0 in/s)
Peak Power = 1.31 kW/m (0.4 kW/ft)
Initial Temperature = 760.0 C (1400 F)
Inlet Subcooling = 11.1 C (20 F)
Figure B-3. Temperature and First Derivative of Temperature Curve at Higher Elevation, Experiment 1096

- Temperature
- First Derivative

Parameters:
- Pressure: 137.9 kPa (20 psia)
- Inlet Flooding: 0.0254 m/s (1.0 in/s)
- Peak Power: 1.31 kW/m (0.4 kW/ft)
- Initial Temperature: 760.0°C (1400°F)
- Inlet Subcooling: 11.1°C (20°F)
Figure B-4. Temperature and First Derivative of Temperature Curve, Experiment 1383

- Temperature (C)
- Time (sec)

P = 275.8 kPa (40 psia)
Inlet Flooding = 0.0254 m/s (1.0 in/s)
Peak Power = 1.31 kW/m (0.4 kW/ft)
Initial Temperature = 760.0 C (1400 F)
Inlet Subcooling = 11.1 C (20 F)
Figure B-5. Quench Temperature at Different Quench Locations, Experiment 1096

- Average temperature = 393.75°C = 740.75°F
- Standard deviation = 39.52°C
- Pressure = 137.9 kPa (20 psia)
- Inlet Flooding = 0.0254 m/s (1.0 in/s)
- Peak Power = 1.31 kW/m (0.4 kW/ft)
- Initial Temperature = 760.0°C (1400°F)
- Inlet Subcooling = 11.1°C (20°F)
Figure B-6. Quench Temperature at Different Quench Locations, Experiment 1383

P = 275.8 kPa (40 psia)
Inlet Flooding = 0.0254 m/s (1.0 in/s)
Peak Power = 1.31 kW/m (0.4 kW/ft)
Initial Temperature = 760.0 C (1400 F)
Inlet Subcooling = 11.1 C (20 F)

average temperature = 420.5 C = 788.9 F
standard deviation = 22.9 C
VITA

Sule Ergun

Sule Ergun was born on June 14, 1974, in Nigde, Turkey. She attended Bahcelievler High School in Ankara, Turkey which she graduated as an honor student in 1991. She attended the Hacettepe University receiving her Bachelor of Science and Engineering degree in Nuclear Engineering. Following graduation she enrolled as a graduate student at the Hacettepe University and earned a M.S. degree in Nuclear Engineering. Her thesis for the M.S. degree detailed probabilistic risk assessment of ITU Triga Mark II research reactor in Istanbul, Turkey. Her work in past 4 years has focused on two phase modeling and experiments. This work included modeling of Dispersed Flow Film Boiling and is described in her Ph.D. dissertation.