CHAPTER III
EXPERIMENTAL

In this chapter, The equipment, test cell was introduced and the experimental methodology was overviewed. Moreover, the mathematical and numerical analyses were also mentioned.

3.1 Materials

The tap water was used as the water flowing through the experimental system and stagnant gas contained in the test cell was the ambient air.

All experiments, the tap water and the ambient air were the supply in Fredericton, Canada.

3.2 Equipment

The test cell was fabricated from a small section of a vertical flux tube assembly identical to that used in the reactor.

From Figure 3.1, it is a diagram of the test cell used in the experiment with the location of the thermocouple. The test cell was designed with the following considerations:

1. It could simulate the conditions to which the detector assembly is exposed in the nuclear reactor moderator.
2. It could be durable for varying the position of strap, thermocouples and heaters, and alteration of water flow rate and generation heating.

3. Sufficient data could be collected to evaluate the ability of the thermocouples located within the detector wells in the reactor detector capsule.

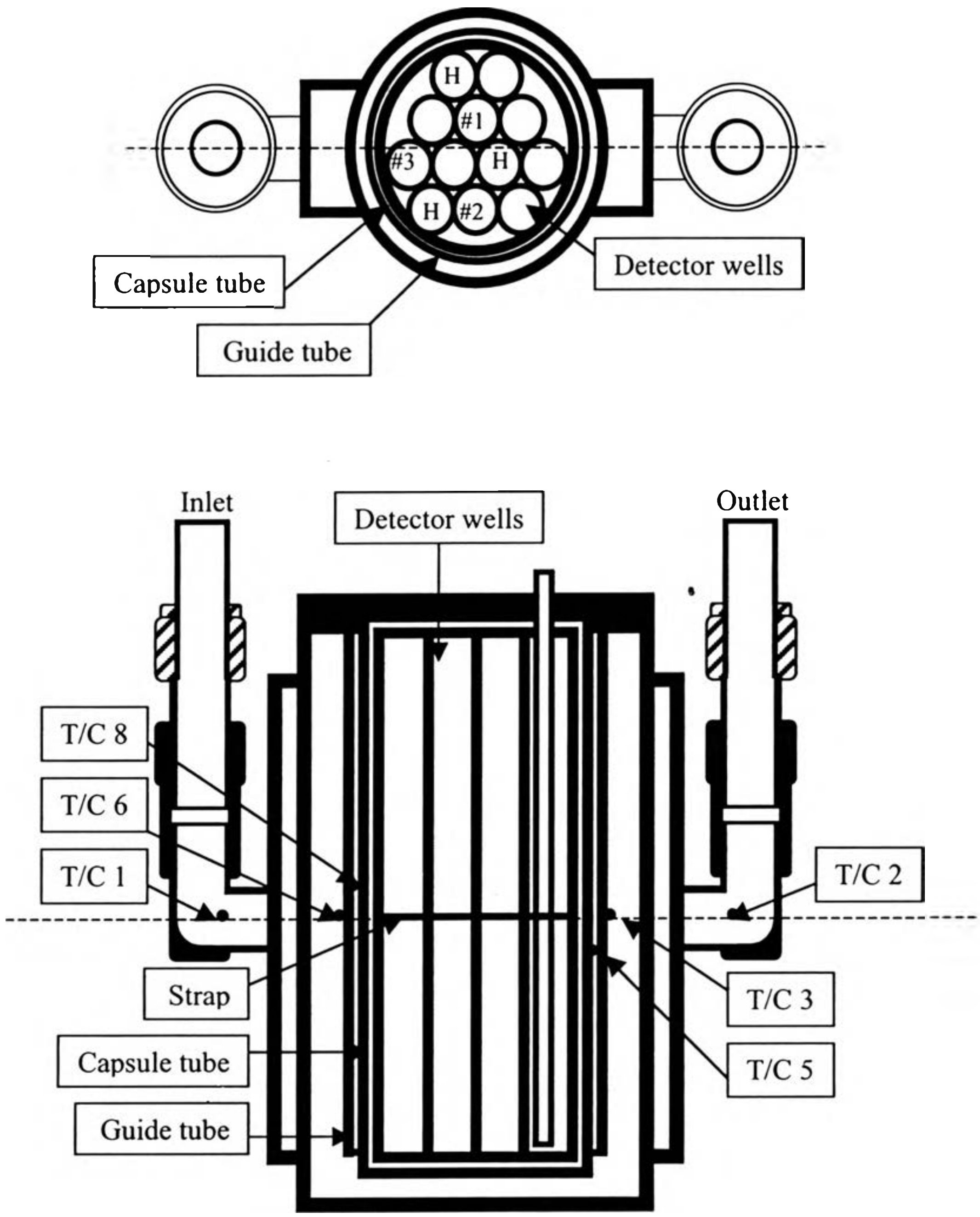


Figure 3.1 Diagram of test cell showing thermocouple placements, H = position of heater, #1, #2, #3 = position of thermocouple of straw 1, 2 and 3, respectively and T/C = position of thermocouple

For the structure of the test cell, the test cell has a length of 6.0 cm. The test section of the test cell was constructed of zircalloy metal. The test section of the test cell was concentrically consisted of the guide tube, the detector capsule and the bundle of the detector wells. All of the compositions and their contents were exactly the same as that assembly used in the moderator of the reactor. In the test cell, there is the strap to wrap around the detector wells and keep them in position. This single strap, which made from zircalloy metal, played an important role in both radial and axial heat transfers. The guide tube was left open at the bottom as in the reactor. A gas inlet line was placed inside one of the detector wells in order to test the system in an atmosphere of helium or air. Surrounding the guide tube is a cavity. The inlet and outlet of this cavity consisted of 45 holes of 2-mm diameter in 3 rows. This part of the test cell called the surrounding thimble tube was fabricated from a stainless steel cylinder along with the inlet and outlet cavities and fittings. Water may be allowed to pass through the guide tube by this cavity to represents the moderator flow over the tube assembly in nuclear reactor. The dimensions and materials of test cell are shown in Table 3.1.

The thermocouples are the devices for measuring temperature profile of the system, which are located for continuously at various positions:

1. Inlet water
2. Outlet water
3. Outside guide tube surface
4. Inside capsule tube surface
5. Inside detector wells for the three different geometric locations

The various positions of thermocouples, as shown in Figure 3.1, were necessary to determine the heat transfer characteristics of the vertical flux tube assembly.

By a calculation, the test cell masses weigh 39.72 grams of zirconium. The estimated maximum irradiative heat in the reactor core of 1.11 watts/gram gives a maximum heat release within the test cell of 44.1 watts in order to give the same heating rate as produced within the center of the reactor core of nuclear reactor (Steward, 2000).

Table 3.1 Dimensions of test cell

Test cell components	Outside diameter (mm)	Inside diameter (mm)	Material
Guide Tube	20.38	18.78	Zircalloy metal
Detector Capsule	16.51	15.5	Zircalloy metal
Detector Wells	3.835	3.429	Zircalloy metal
Surrounding Thimble Tube	48.26	38.1	Stainless steel
Holes	Number	Diameter (mm)	Rows
Inlet	45	2	3
Exit	45	2	3
Detector Strap	Width 3.175	Thickness 0.127	Zircalloy metal
Length of cell	60 mm		

3.3 Experimental Methods

3.3.1 Temperature Distribution Study in Steady State Condition

It may be possible to measure the temperature distribution of reactor moderator by modeling the test cell and installing thermocouples as temperature recording devices within flux tubes of test cell. The apparatus for this set of experiments is shown in Figure 3.2.

For the scheme of steady state experiments, the test cell was connected to the water reservoir and an electrical source, which generated the simulated irradiative heating within flux assembly. The heat was passed through the test cell by three electrical heaters, which were inserted in the test cell detector wells as shown in Figure 3.1. These heaters can yield 6.9 watts/cm. Therefore, three inserted heaters can produce the level of internal heating as the irradiation within the center of the nuclear reactor core. The series of tests used three electrical heaters at various levels of heating under the flow conditions expected within the moderator. In the detail of the experiment procedure, water at fixed temperature from the temperature-controlled reservoir was allowed to constantly flow through the test cell by using a pump. The flow of water was controlled by the flow meter, a rotameter, until thermal equilibrium was achieved. After the equilibration of temperature was achieved, the electrical source was switched to the level of heating desired. The thermocouples were used to record the temperature until thermal equilibrium was achieved.

All experimental runs were preformed with water flowing through the test cell at room temperature, about 30°C.

The examined parameters were water flow rate, rate of heating, the position of strap, the position of heater, and the position of thermocouples inserted within the test cell. These were considered to be the variable parameters effecting the moderator temperature distribution. The water flow rate was varied from 1.5 to 7.6 USGPM (equality of the water velocity from 0.14 to 0.96 m/s). The rate of heating was various from 1.08 to 17.5 watts. The position of thermocouple within the test cell was varied every 1 cm from 1.5 to 6.5 cm away from the top of test cell cap. The positions of three electrical heaters were at the center and at the bottom of the test cell. The position of the strap was considered in 3 positions; 2.5, 3.5 and 4.5 cm away from the top of test cell cap.

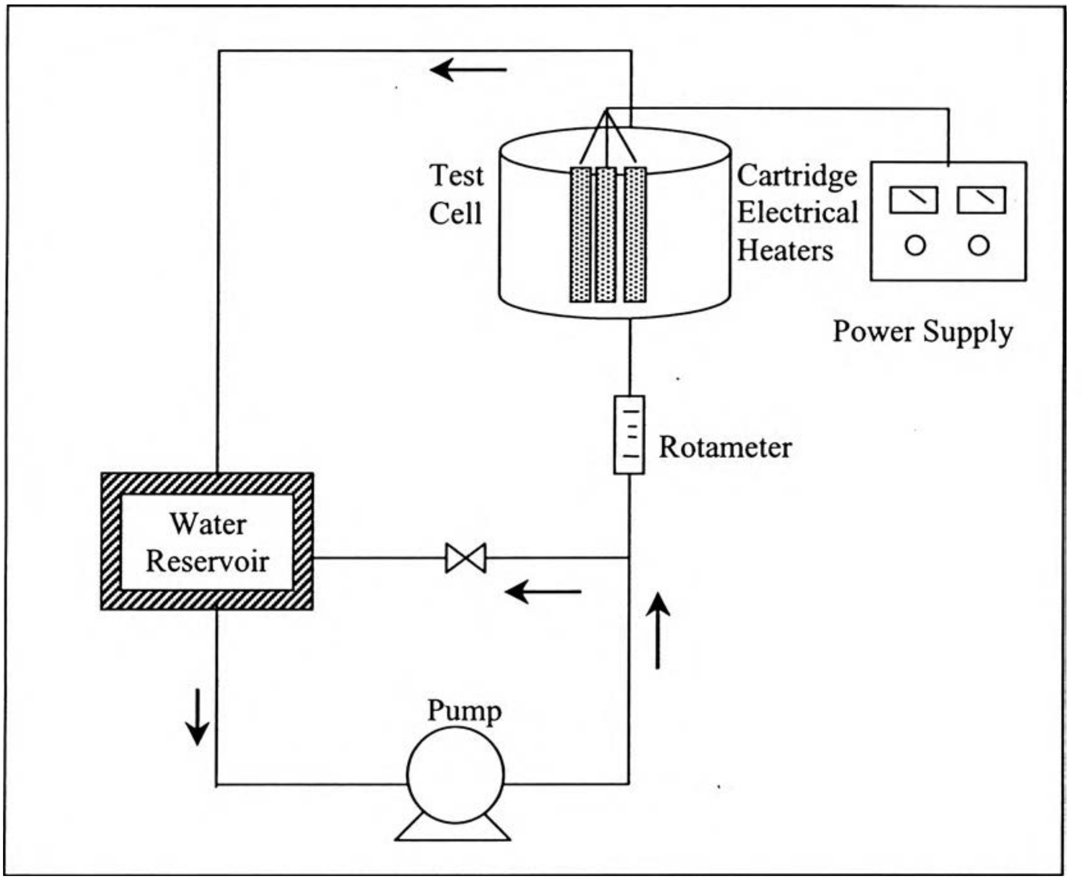


Figure 3.2 Experimental arrangement for steady state tests using the electrical heaters

3.4 Mathematical Analysis

3.4.1 One Dimensional Steady State Analysis

3.4.1.1 For Radial Heat Transfer

A method of considering the heat transfer characteristics of the flux tube assembly was to perform a steady state experiment on the test cell generated heating by electrical heater. This test procedure was described for the heat transfer in the radial direction.

This analysis assumed the model shown in Figure 3.3 could be represented by concentrical components of flux assembly.

The model assumed that the mass of the flux assembly was divided into three parts, an outer tube representing the guide tube, an inner tube representing the detector capsule tube and a center cylinder representing the detector wells. In this model, the center cylinder part was modified from the bundle shape of the actual detector tubes to the ring shape.

The equations and boundary conditions were presented for steady state heating of the mass within the flux detector assembly.

The steady state energy equations for the outside tube, the inner tube and the center ring were performed for two cases, irradiative heating and electrical heating.

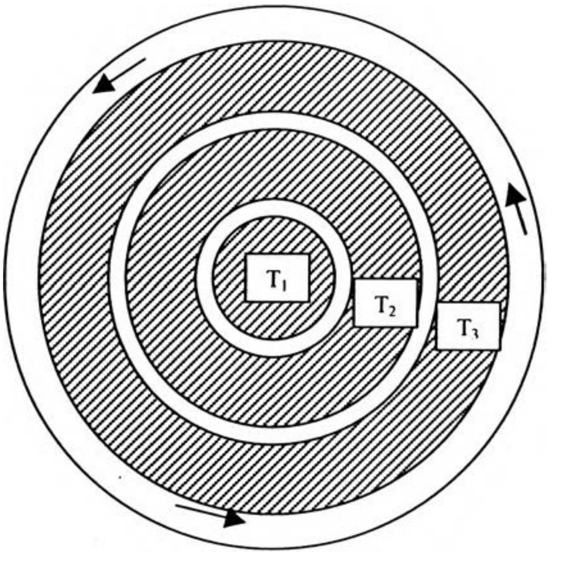


Figure 3.3 Representation of VFD for steady state analysis based on the radial heat transfer

3.4.1.1.1 For Electrical Heating

In the case of electrical heating, only center cylinder has heat generation as indicated below.

Outside tube

$$\frac{d}{dr}\left(r\frac{dT_3}{dr}\right)=0 \tag{3.1}$$

Inner tube

$$\frac{d}{dr}\left(r\frac{dT_2}{dr}\right)=0 \tag{3.2}$$

Center cylinder

$$\frac{d}{dr}\left(r\frac{dT_i}{dr}\right)=-\frac{qr}{k} \tag{3.3}$$

The boundary conditions based on internal electrical heating of the mass within the flux tube assembly are as same as the ones of irradiative heating.

The solution to these equations and boundary conditions is presented in Appendix A.2. In an approximate solution, the major resistance to heat transfer was between the inner tube and the central cylinder, and using the physical and geometric properties of the flux assembly is given by

$$T_1-T_s=q\left(\frac{0.1485}{h_1}+\frac{0.1394}{h_2}+\frac{0.12847}{h_3}+0.1763\right) \tag{3.4}$$

where: q is the heat generation per unit volume of the center cylinder, W/cm°C.

k is the thermal conductivity of the center cylinder, inner tube and outer tube, W/cm°C.

T_1 is the temperature within the center cylinder, °C.

T_2 is the temperature within the inner tube, °C.

T_3 is the temperature within the outer tube, °C.

T_s is the temperature of flowing fluid surrounding the outer tube, °C.

h_1 is the heat transfer coefficient between the center cylinder and the inner tube, W/cm^2°C.

h_2 is the heat transfer coefficient between the inner tube and the outer tube, W/cm^2°C.

h_3 is the heat transfer coefficient between the outer tube and the water flowing over the outer tube, W/cm^2°C.

r is the distance from the center of the model, cm.

The steady state energy equation for irradiative heating (gamma cell) is shown in Appendix A.1

For axial heat transfer that the heat transfer behavior of the detector assembly is considered in the axial flow direction by using the one-dimensional geometry of steady state condition with heat generation at the center of the test cell is shown in Appendix A.3.

In addition to a one-dimensional analysis, the detector assembly was concerned in a two-dimensional analysis since the experimental data obtained in this study revealed that the heat transfer behavior for the heated assembly depended on the radial and axial dimensions. This is due primarily to the straps, which hold the bundle of the flux tubes inside the flux detector capsules. At the strap, some part, it has good contract with the flux detector capsule, which improves the heat transfer to the capsule tube, whereas the flux tubes do not touched with the flux detector capsule tube. At the strap, the resistance to heat transfer between the center cylinder representing the flux tubes and the flux detector capsule is significantly reduced. Even if the two-dimensional equations with the boundary conditions can not be solved analytically, however, the results for these cases can be obtained by using numerical solutions such as the computational heat transfer program named FLUENT.

3.5 Numerical Analysis

3.5.1 Introduction to Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is the analysis by numerical computation. The performance of mathematical-code calculations resembles the performance of physical experiments for fluid flow, heat transfer and associated multifarious transport phenomenon such as chemical reactions. CFD is viewed as the 'third dimension' in fluid dynamics analysis, which is in conjunction with the 'pure experiments', gives practical results as in Figure 3.4 (Malalasekera and Versteeg, 1995).

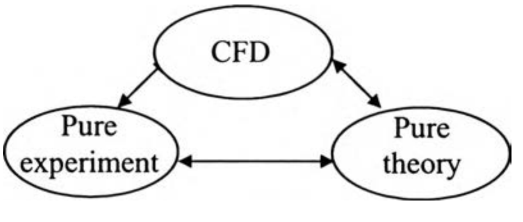


Figure 3.4 Relationship between pure experiment and pure theory

This technique is very powerful and resourceful in a wide range of industrial and non-industrial applications since the code allows changing operational parameters and conditions without human risk or obstruction, and supports the accurate results in a cost effective manner.The cornerstone of Computational Fluid Dynamic is based on three fundamental physical principles. These are expressed in the form of the governing equations of fluid dynamics: continuity, momentum and energy equations.

CFD codes are structured to solve the numerical algorithms. For the structure of the CFD, all of the commercial CFD packages can be managed into three main steps of procedure in order to provide easy access (Roache, 1972):

- pre-processor
- solver
- post-processor

The pre-processor is the element, which consists of the flow problem input. It concern about defining the geometry of the interesting system. Pre-processor is an important and time-consuming part in order to make an accurate and robust fundamental of CFD solution.

The solver combines the physical models and the flow conditions. The bases of these numerical solution techniques perform in steps.

The post-processor has been utilized to support an amount of development work including the outstanding graphics. It presents many of the versatile data implements. This element helps the user to better understand and visualize the solution.

This program uses the principle of iterations and three mathematical concepts: convergence, consistency and stability, to determine the success of algorithms.

There are many commercial computer codes for different analysis and versatile application: FULENT, Tacs-FLOW, CERCA, STAR-CD, etc.

In this thesis, The software named FLUENT, from FLUENT Inc. was used.

3.5.2 Introduction of FLUENT

FLUENT is the flexible and powerful software written in the C computer language. As other CFD codes, this software can solve the dynamic behavior of the flow. It can be applied for the problem of aircraft industry, chemical industry, and automotive industry, etc.

FLUENT is suited for modeling fluid flow, heat and mass transfer in complex geometry, chemical reactions, and relative phenomena. FLUENT provides complete mesh flexibility, solving your flow problems with unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D-triangular/ quadrilateral, 3D tetrahedral/ hexahedral/ pyramid/ wedge, and mixed (hybrid) meshes. FLUENT also allows you to refine or coarsen your grid based on the flow solution.

This solution-adaptive grid capability is very useful for accurately predicting flow fields in regions with large gradients, such as free shear layer and boundary layers. The FLUENT package includes main and additional pre-processors and main solvers, FLUENT that each solver has its own post-processing feature.

- Pre-processor: GeoMesh and Gambit

GeoMesh is come from the geometry and mesh generation preprocessor. It purposes to be a pre-processor program providing the functionality that is needed to create analysis models based on given geometry, and to generate meshes for the FLUENT CFD solvers. GeoMesh uses the geometry modeler, ICEM DDN, in order to create and/or modify geometry and uses the unified block modeler/grid generator called GeoMesh P-Cube to create a set of blocks activating grid generation functions, and then to automatically generate the meshes. Besides, in order to visualize and diagnose the quality of the mesh, GeoMesh uses the grid visualizer, GeoMesh Leo.

GAMBIT is also a software package functioning as the pre-processor, which was designed to help analysts and designers to build and mesh models for applying with FLUENT. GAMBIT is versatile to accommodate a wide range of modeling applications.

– *Solver: FLUENT 5*

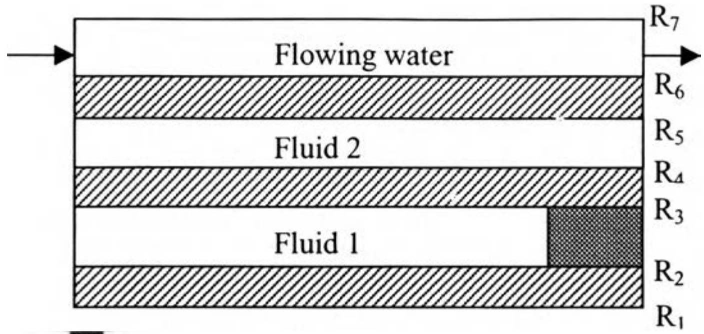
The solver software applied in this study due to the heat transfer condition of flux tube assembly is FLUENT 5. It is suitable for compressible and incompressible flows, and able to apply with the flow involving complex geometries and complex physics.

– Problem Solving Steps in CFD

1. Create the model geometry and grid by using GeoMesh or GAMBIT, and then import into FLUENT.
2. Start the appropriate solver, FLUENT 5, for 2D or 3D modeling.
3. Read the grid file and check it in order to see the domain extents, volume statistics, and connectivity information.
4. Select the solver formulation that is provided in three different forms in FLUENT: segregated, coupled implicit, and coupled explicit.
5. Choose the basic equations to be solved: laminar or turbulent, chemical species or reaction, heat transfer models, etc. Identify additional models needed: fans, heat exchangers, porous media, etc.
6. Specify material properties involving in the work.
7. Specify the boundary conditions for the model.
8. Adjust the solution control parameters.

9. Initialize the flow field to provide a starting point for the solution before iterating.
10. Calculate a solution.
11. Examine the results.
12. Save the results. The problem and the results computed by FLUENT are stored in two separate files: the case file and the data file.
13. If necessary, refine the grid or consider revisions to the numerical or physical model.

In the simulation calculation, the transfer of heat was considered in a two-dimension analysis with steady state condition with two cases of the heat source, irradiative heating (gamma) and electrical heating as shown the equations in Appendices A.4 and A.5. For the irradiative heating case, the heat generation was in all solid phases and fluid 1. For the electrical heating, there was the heat generation at the center of the bundle of detector wells or the center cylinder of the model. For the initial conditions, system temperature was at 300K. Water velocity was at 0.96 m/s. Heating rates were at 5, and 10 W. In the FLUENT model, there was the mock-up of half of the test cell due to symmetry as shown in Figure 3.5.



Representing the strap:

Representing solid phase: zirconium.

Representing Fluid phase:

Fluid 1 is varied material: air, helium etc.

Fluid 2 is stagnant water.

Fluid 3 is flowing water.

Figure 3.5 The model of test cell used in FLUENT 5

The strap wrapped around the detector tubes was in contact touched at some points of the detector capsule tube, approximately 15 percent of the total surface area of the capsule tube. For the model, the whole strap is allowed to touch the solid phase representing the detector capsule tube. However, the thermal conductivity of the strap was set equal to 0.15 of the thermal conductivity of the detector capsule.

All of the solid layers including of the strap metal were zirconium. Fluid layers were separated into 3 parts, 2 liquid parts and 1 gas phase part. The top part was flowing liquid, which was acted as the moderator of the system, was water (H_2O). The middle fluid layer was the stagnant liquid, which is also water. The bottom layer was the gas phase located close to the center cylinder and inner tube. It was a gas fed into the assembly. The material for this layer was a variable parameter, either air or helium. For the bottom metal, it used a ring shape to create the simulation model of the bundle of the flux detector tubes.

This model was applied to study the trend of heat transfer from the bottom part to the top part of the flux tube assembly. The significant point of this calculation was the zirconium strap, which significantly affected the heat transfer within the assembly.