Thermal-hydraulic simulation and analysis of
Research Reactor Cooling Systems

By
HESHAM HASANAIN ALY EL KHATIB

A Thesis Submitted in Partial Fulfillment of the Requirements for the Degree
of Philosophy of Doctoral

In
MECHANICAL ENGINEERING

Under the Supervision of

Prof. Dr. Karam M. H. El Shazly
Prof. Dr. Maher G. A. Higazy
Dept. of Mech. Engineering
Dept. of Mech. Engineering
Benha University
Benha University

Prof. Dr. Salah El-Din B. El-Morshedy
Reactors Department
Atomic Energy Authority

FACULTY OF ENGINEERING AT SHOUBRA
BENHA UNIVERSITY
2013
TABLE OF CONTENTS

CONTENTS

List of figures
List of tables
List of symbols
Acknowledgment
Abstract
List of publications

CHAPTER (1)

Introduction

1.1 Research Reactors
1.2 Types of research reactors
1.3 Uses of research reactors
1.4 Cooling description
1.4.1 Cooling circuits
1.4.2 Core cooling system
1.4.3 Pool cooling system
1.4.4 Secondary cooling system
1.5 Heat exchanger in MTR
1.6 Cooling tower in MTR
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.7</td>
<td>Postulated initiating events and accident scenarios potentially leading to core damage in research reactors</td>
<td>19</td>
</tr>
<tr>
<td>1.7.1</td>
<td>Power excursion due to insertion of excess reactivity</td>
<td>20</td>
</tr>
<tr>
<td>1.7.2</td>
<td>Loss of flow accident (LOFA)</td>
<td>21</td>
</tr>
<tr>
<td>1.7.3</td>
<td>Loss of coolant accident (LOCA)</td>
<td>23</td>
</tr>
<tr>
<td>1.7.4</td>
<td>Fuel handling accidents</td>
<td>24</td>
</tr>
<tr>
<td>1.7.5</td>
<td>Flow redistribution condition</td>
<td>25</td>
</tr>
<tr>
<td>1.8</td>
<td>Power peaking factor related design basis</td>
<td>26</td>
</tr>
<tr>
<td>1.9</td>
<td>Safety relevant design basis</td>
<td>26</td>
</tr>
<tr>
<td>1.9.1</td>
<td>Critical phenomena</td>
<td>26</td>
</tr>
<tr>
<td>1.9.1.1</td>
<td>Departure from nucleate boiling.</td>
<td>27</td>
</tr>
<tr>
<td>1.9.1.2</td>
<td>Flow instability phenomena.</td>
<td>27</td>
</tr>
<tr>
<td>1.9.1.3</td>
<td>Low flow burn-out</td>
<td>28</td>
</tr>
<tr>
<td>1.9.1.4</td>
<td>Fluid-structure interactions</td>
<td>28</td>
</tr>
<tr>
<td>1.10</td>
<td>Non-safety relevant design basis</td>
<td>29</td>
</tr>
<tr>
<td>1.11</td>
<td>Objectives</td>
<td>29</td>
</tr>
</tbody>
</table>

**CHAPTER (2)**

**Literature review**

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Critical phenomena limiting research reactor power</td>
<td>32</td>
</tr>
<tr>
<td>2.2</td>
<td>Thermal-hydraulic analysis and simulation in MTR</td>
<td>36</td>
</tr>
<tr>
<td>2.3</td>
<td>Heat exchangers and cooling towers review</td>
<td>59</td>
</tr>
</tbody>
</table>
CHAPTER (3)

Mathematical model

3.1 Distribution of heat generation in the fuel core
3.2 Reactor core model
3.3 Steady state analysis
3.3.1 Fuel analysis
3.3.2 Fuel temperature calculation
3.3.3 Clad temperature calculation
3.3.3.1 Outer surface temperature
3.3.3.2 Inner surface temperature
3.3.4 Coolant temperature
3.4 Calculation of the heat transfer coefficient
3.5 Transient-state
3.5.1 Fuel temperature
3.5.2 Aluminum clad analysis
3.5.3 Coolant analysis
3.6 Heat exchanger model
3.7 Cooling tower analysis
3.8 Pipe line analysis
3.9 Thermal-hydraulic safety margins
3.9.1 Onset of Nucleate Boiling
3.9.2 Onset of flow instability
3.9.3 Departure from nucleate boiling 93
3.9.4 Determination of hydraulic stability of the fuel plate. 94
3.10 Two-dimensional steady state of fuel plate 95

CHAPTER (4)

Model assessment 99
4.1 Steady-state 99
4.2 Transient-state 101

CHAPTER (5)

Results and discussions 107
5.1 Core heat flux profile at design condition 108
5.2 Reactor core steady state 108
5.2.1 Core temperature profile at design condition 110
5.3 Reactor start-up simulation 111
5.3.1 Regime (I) simulation 111
5.3.2 Regime (II) simulation 114
5.4 Loss of heat sink simulation 117
5.4.1 Regime I simulation 117
5.4.2 Regime II simulation 123
5.5 Thermal-hydraulic safety margins 127
5.6 Hydraulic stability of fuel plate 128
5.7 Clad-surface temperature contour 129
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Conclusions and recommendations</td>
<td>133</td>
</tr>
<tr>
<td>6.1 Conclusions</td>
<td>133</td>
</tr>
<tr>
<td>6.2 Recommendation of future work</td>
<td>135</td>
</tr>
<tr>
<td>References</td>
<td>136</td>
</tr>
<tr>
<td>APPENDIX (1)</td>
<td>144</td>
</tr>
<tr>
<td>Program model in EES</td>
<td>144</td>
</tr>
<tr>
<td>APPENDIX (2)</td>
<td>167</td>
</tr>
<tr>
<td>Decay power</td>
<td>167</td>
</tr>
</tbody>
</table>
## List of Figures

| Figure (1-1) | Basic core arrangement. | 9  |
| Figure (1-2) | Fuel plate assembly.     | 10 |
| Figure (1-3) | Natural circulation flow path. | 13 |
| Figure (1-4) | Core thermal hydraulic.  | 14 |
| Figure (1-5) | Reactor core cooling system. | 14 |
| Figure (1-6) | Pool cooling system.     | 15 |
| Figure (1-7) | Secondary cooling system. | 15 |
| Figure (1-8) | Exploded View of Plate Type Heat Exchange. | 17 |
| Figure (1-9) | Cooling tower cell.      | 18 |
| Figure (3-1) | View of the unit cell representing the MTR fuel plate. | 71 |
| Figure (3-2) | Fuel plate control volume. | 79 |
| Figure (3-3) | Control volume of heat exchanger. | 85 |
| Figure (3-4) | Control volume of cooling tower. | 88 |
| Figure (3-5) | Pipe line control volume. | 90 |
| Figure (3-6) | Nodal network of two dimensional conduction in fuel plate | 95 |
| Figure (4-1) | Temperature profile along the average channel. | 100 |
| Figure (4-2) | Temperature profile along the hot channel. | 100 |
| Figure (4-3) | Core inlet and outlet temperatures. | 102 |
| Figure (4-4) | Cooling tower inlet and outlet temperatures. | 103 |
| Figure (4-5) | Heat exchanger inlet and outlet temperatures. | 103 |
| Figure (4-6) | Cooling tower outlet temperatures. | 104 |
| Figure (4-7) | Core inlet and outlet temperatures. | 105 |
| Figure (5-1) | Flux distribution in average and hot channel. | 107 |
| Figure (5-2) | Temperature profile along the average channel. | 109 |
| Figure (5-3) | Temperature profile along the hot channel. | 109 |
| Figure (5-4) | Temperature distribution at the fuel plate center. | 111 |
| Figure (5-5) | Maximum hot channel temperatures during start-up for regime I. | 112 |
| Figure (5-6) | Maximum temperatures during start-up for regime I. | 112 |
| Figure (5-7) | Core inlet and outlet temperatures in regime I. | 113 |
| Figure (5-8) | Maximum hot channel temperatures during start-up for regime II. | 114 |
| Figure (5-9) | Maximum hot channel temperatures during start-up for regime II. | 115 |
| Figure (5-10) | Core inlet and outlet temperatures in regime II. | 116 |
| Figure (5-11) | Cooling tower inlet and outlet temperature in regime I and II. | 116 |
| Figure (5-12) | Cooling tower outlet temperature in regime I. | 120 |
| Figure (5-13) | Inlet coolant temperature variation during loss of heat sink for regime I. | 121 |
| Figure (5-14) | Maximum coolant temperature during loss of heat sink for regime I. | 121 |
Figure (5-15)  Maximum clad-surface temperature variation during loss of heat sink for regime I.  122
Figure (5-16)  Maximum fuel-center temperature variation during loss of heat sink for regime I.  122
Figure (5-17)  Cooling tower outlet temperature in regime II.  125
Figure (5-18)  Core inlet temperature variation during loss of heat sink for regime II.  125
Figure (5-19)  Maximum coolant temperature variation during loss of heat sink for regime II.  126
Figure (5-20)  Maximum clad-surface temperature variation during loss of heat sink for regime II.  126
Figure (5-21)  Maximum fuel-center temperature variation during loss of heat sink for regime II.  127
Figure (5-22)  Temperature contour of clad surface in average channel regime I.  129
Figure (5-23)  Temperature contour of clad surface in hot channel regime II.  130
Figure (5-24)  Temperature contour of clad surface in abnormal regime I.  131
Figure (5-25)  Temperature contour of clad surface in abnormal regime II.  132
# List of tables

<table>
<thead>
<tr>
<th>Page</th>
<th>No.</th>
<th>Table [1.1]</th>
<th>ETRR-2 design parameters.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>7</td>
<td>Table [1.2]</td>
<td>Plate heat exchanger specifications.</td>
</tr>
<tr>
<td></td>
<td>17</td>
<td>Table [1.3]</td>
<td>Cooling tower specifications.</td>
</tr>
<tr>
<td></td>
<td>18</td>
<td>Table [4.1]</td>
<td>Maximum temperature values predicted by the present model and PARET code.</td>
</tr>
<tr>
<td></td>
<td>101</td>
<td>Table [5.1]</td>
<td>Maximum temperature values predicted by the present model in regime I.</td>
</tr>
<tr>
<td></td>
<td>110</td>
<td>Table [5.2]</td>
<td>Maximum temperature values predicted during loss of heat sink for regime I.</td>
</tr>
<tr>
<td></td>
<td>119</td>
<td>Table [5.3]</td>
<td>Maximum temperature values of tower during loss of heat sink for regime I.</td>
</tr>
<tr>
<td></td>
<td>120</td>
<td>Table [5.4]</td>
<td>Maximum temperature values predicted during loss of heat sink for regime II.</td>
</tr>
<tr>
<td></td>
<td>124</td>
<td>Table [5.5]</td>
<td>Best-estimate minimum safety margins.</td>
</tr>
<tr>
<td></td>
<td>128</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
## List of symbols and abbreviations

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Cross sectional area of the coolant channel,</td>
</tr>
<tr>
<td>(A_{ps})</td>
<td>Convective surface area,</td>
</tr>
<tr>
<td>b</td>
<td>Fuel plate width,</td>
</tr>
<tr>
<td>Bi</td>
<td>Biot number,</td>
</tr>
<tr>
<td>c</td>
<td>Constant which depends on support condition for fuel plate,</td>
</tr>
<tr>
<td>CHF</td>
<td>Critical heat flux,</td>
</tr>
<tr>
<td>(c_p)</td>
<td>Specific heat of the coolant,</td>
</tr>
<tr>
<td>CCFL</td>
<td>Counter current flow limitation,</td>
</tr>
<tr>
<td>d</td>
<td>Half width of channel,</td>
</tr>
<tr>
<td>(D_h)</td>
<td>Hydraulic diameter of coolant channel,</td>
</tr>
<tr>
<td>E</td>
<td>Energy,</td>
</tr>
<tr>
<td>H</td>
<td>Enthalpy,</td>
</tr>
<tr>
<td>(h_{conv})</td>
<td>Convective heat transfer coefficient,</td>
</tr>
<tr>
<td>HEU</td>
<td>High Enriched Uranium,</td>
</tr>
<tr>
<td>HFIR</td>
<td>High Flux Isotope Reactor,</td>
</tr>
<tr>
<td>ILL</td>
<td>The Institute Laue Laugvine,</td>
</tr>
<tr>
<td>L</td>
<td>Pipe length,</td>
</tr>
<tr>
<td>(L_c)</td>
<td>Characteristic length,</td>
</tr>
<tr>
<td>LEU</td>
<td>Low Enriched Uranium,</td>
</tr>
<tr>
<td>MTR</td>
<td>Material testing reactor</td>
</tr>
<tr>
<td>M</td>
<td>Mass,</td>
</tr>
<tr>
<td>w</td>
<td>Mass flow rate,</td>
</tr>
<tr>
<td>W</td>
<td>Reactor thermal power,</td>
</tr>
<tr>
<td>K</td>
<td>Conductive heat transfer coefficient,</td>
</tr>
<tr>
<td>Nu</td>
<td>Nusselt number,</td>
</tr>
<tr>
<td>ONB</td>
<td>Onset of nucleate boiling,</td>
</tr>
<tr>
<td>OFI</td>
<td>Onset of flow instability,</td>
</tr>
<tr>
<td>DNB</td>
<td>Departure from nucleate boiling,</td>
</tr>
</tbody>
</table>
Pr  Prandtl number,
per  Perimeter of the coolant channel,
P  Local pressure
PPF  Power peaking factor.
q  Transferred heat by conduction or convection,
Re  Reynolds number,
t  Channel thickness,
T  Temperature,
U  Overall heat transfer coefficient,
v  Average heat transfer coefficient in centerline of coolant channel,
V_f  Fuel plate volume,
x  Direction of thickness,
z  Direction of height,
z_c  Semi-active height of fuel plate,
z_e  Extrapolated length,

Greek letters
ρ  Coolant density,
μ  Dynamic viscosity of the coolant,
γ  Kinematic viscosity of coolant,
φ  Thermal heat flux,
η  Bubble detachment parameter

Subscription
a  Air,
c.v  Control volume,
cl  Clad,
conv.  Convection,
e  Exit,
f  Fuel,
FE  Fuel element,
FP  Fuel plate,
g Generated,
h Heated.
hx Heat exchanger,
in Inlet,
LMTD Logarithmic mean temperature difference,
m Maximum,
out Outlet,
p Pipe,
r Removed,
sat Saturated,
w Water.
ACKNOWLEDGMENT

All praises and thanks are due to Allah (subhana wa taala) for bestowing me with health, knowledge and patience to complete this work. I wish to express my sincere gratitude and appreciation first to Prof. Dr. Salah-El Din El-Morshedy; Atomic Energy Authority, Prof. Dr. Karam Mahmoud El Shazly Faculty of engineering Benha university and Prof. Dr. Maher Higazy; Faculty of engineering, Benha university. For their supervision, cooperation, assistance and active guidance during the preparation of this thesis

Thanks are also extended to my colleagues in Atomic Energy Authority-Egyptian second research reactor (ETRR-2) for their great assistance and providing me with complete operational records to finish this work.

Finally, thanks are due to my dearest mother and late father, and all my family members for their emotional and moral support throughout my academic career and also for their love, patience, encouragement and prayers.
ABSTRACT

The objective of the present study is to formulate a model to simulate the thermal hydraulic behavior of integrated cooling system in a typical material testing reactor (MTR) under loss of ultimate heat sink, the model involves three interactively coupled sub-models for reactor core, heat exchanger and cooling tower. The developed model predicts the temperature profiles in addition it predicts inlet and outlet temperatures of the hot and cold stream as well as the heat exchangers and cooling tower. The model is validated against PARET code for steady-state operation and also verified by the reactor operational records, and then the model is used to simulate the thermal-hydraulic behavior of the reactor under a loss of ultimate heat sink. The simulation is performed for two operational regimes named regime I of (11MW) thermal power and three operated cooling tower cells and regime II of (22MW) thermal power and six operated cooling tower cells. In regime I, the simulation is performed for 1, 2 and 3 cooling tower failed cells while in regime II, it is performed for 1, 2, 3, 4, 5 and 6 cooling tower failed cells. The safety action is conducted by the reactor protection system (RPS) named power reduction safety action, it is triggered to decrease the reactor power by amount of 20% of the present power when the water inlet temperature to the core reaches 43°C and a scram (emergency shutdown) is triggered in case of the inlet temperature reaches 44°C. The model results are analyzed and discussed. The temperature profiles of fuel, clad and coolant are predicted during transient where its maximum values are far from thermal hydraulic limits.
CHAPTER (1)

INTRODUCTION

There are many nuclear research reactors in different types all over the world. Generally, they are not used for power generation. According to the IAEA [1], 651 nuclear research reactors have been built in 92 countries around the world. Almost 284 research reactors are currently in operation, 258 are shut down and 109 have been decommissioned. The primary purpose of research reactors is to provide a neutron source for research and other purposes, but not for power. However, high power densities are involved in the core, there are many different types of research reactors, and plate type fuel research reactor is one of the most common ones. Because it is impractical and undesirable to test nuclear reactor under accident conditions, complicated numerical computer programs (often called codes) are developed. These codes are directed toward improving the ability to understand specific phenomena and the results of tests, with objective of applying this understanding to the interpretation of results if resembling phenomena were to happen. The result of the development and use of complex digital computer codes is to obtain the time dependence of important parameters during accident sequences. These computer codes attempt to simulate the important physical processes through numerical solution of large systems of differential equations.

Nuclear computer codes are of two general types:
1-The first type is used for steady-state analysis and is called core simulator.
2-The second type is used for the analysis of accidental and operational transients.
This type of codes combines time-dependent thermal-hydraulics and neutron kinetics along with appropriate feedback mechanisms. Such codes are known as
neutronics thermal hydraulics codes, this later type of codes has a wide spectrum of applications. The transients analyzed by these codes may be categorized as either neutronically or thermal hydraulically driven, depending on the initiating event.

During the design, licensing, and perform of this type of reactors, computer codes are generally required. Many codes have been developed for the analysis of some anticipated transients and accidents concerned about nuclear reactors. It is required to perform transient analysis to deal with the review of proposed tests or experiments, almost all the analyses of research reactor transients are carried out with the help of large code systems such as RELAP, RETRAN, CATHARE, ATHLET and PARET that simulate the coupled kinetics and thermal-hydraulics of the reactor core. The use of a large code is sometimes difficult with respect to input preparation and output processing; also, most of the code systems for transient analysis are tailored to power reactors and not well suited for research reactor applications. Although some of these power reactor codes have been adapted for research reactor applications, a simpler code should be sufficient in most purposes.

The fundamental objective of nuclear reactor safety is to protect the public from the radioactive products of fission process. This goal is achieved by design philosophy called defense-in-depth in which three barriers are placed between the radioactive elements and the environment. These barriers levels are the fuel element cladding, the reactor vessel and the containment building. Maintaining the integrity of these three barriers for both normal and abnormal operating conditions is the primary task of nuclear reactor safety engineering. Thermal hydraulic considerations are important when selecting overall plant characteristics. Primary system temperature and pressure are key characteristics related to both the coolant selection and plant thermal performance. This thermal performance is dictated by the bounds of the maximum allowable
primary coolant outlet temperature and the minimum achievable condenser coolant inlet temperature.

1.1 Research Reactors

Many of the world's nuclear reactors are used for research and training, materials testing, or the production of radioisotopes for medicine and industry. They are basically neutron factories. These are much smaller than power reactors or those propelling ships, and many are on university campuses. Some research reactors operate with high-enriched uranium fuel, and international efforts are underway to substitute low-enriched fuel. Some radioisotope production also uses high-enriched uranium as target material for neutrons, and this is being phased out in favor of low-enriched uranium. Research reactors comprise a wide range of civil and commercial nuclear reactors which are generally not used for power generation. The term is used here to include test reactors, which are more powerful than others. The primary purpose of the research reactors is to provide a neutron source for research and other purposes. Their output (neutron beams) can have different characteristics depending on use. They are small relative to power reactors whose primary function is to produce heat to make electricity. They are essentially net energy users; their power is designated in megawatts (or kilowatts) thermal (MW$_{th}$ or MW$_t$), but here we will use simply MW (or kW). Most range up to 100MW, compared with 3000MW (i.e. 1000MW) for a typical power reactor. Research reactors are simpler than power reactors and operate at lower temperatures. They need far less fuel, and far less fission products build up as the fuel is used. On the other hand, their fuel requires more highly enriched uranium, typically up to 20% U-235, although some older ones use 93% U-235. They also have a very high power density in the core, which requires special design features. Like power reactors, the core needs cooling. Though only the higher-powered test reactors need forced cooling. Usually a moderator is required to slow down the
neutrons and enhance fission. As neutron production is their main function, most research reactors also need a reflector to reduce neutron loss from the core.

1.2 Types of research reactors

There is a much wider array of designs in use for research reactors than for power reactors, where 80% of the world's plants are of just two similar types. They also have different operating modes, producing energy which may be steady or pulsed, Colin West [2].

A common design (67 units) is the pool type reactor, where the core is a cluster of fuel elements sitting in a large pool of water. Among the fuel elements are control rods and empty channels for experimental materials. Each element comprises several (e.g. 18) curved aluminum-clad fuel plates in a vertical box. The water both moderates and cools the reactor, and graphite or beryllium is generally used for the reflector, although other materials may also be used. Apertures to access the neutron beams are set in the wall of the pool. Tank type research reactors (32 units) are similar, except that cooling is more active.

The TRIGA reactor is another type and common design (40 units). The core consists of 60-100 cylindrical fuel elements about 36 mm diameter with aluminum cladding enclosing a mixture of uranium fuel and zirconium hydride (as moderator). It sets in a pool of water and generally uses graphite or beryllium as a reflector. This kind of reactor can safely be pulsed to very high power levels for fractions of a second. Its fuel gives the TRIGA a very strong negative temperature coefficient, and the rapid increase in power is quickly cut short by a negative reactivity effect of the hydride moderator.

Other designs are moderated by heavy water (12 units) or graphite. A few are fast reactors, which require no moderator and can use a mixture of uranium and plutonium as fuel. Homogenous type reactors have a core comprising a solution of uranium salts as a liquid, contained in a tank about 300 mm diameter. The simple design made them popular early on, but only five are now operating.
Research reactors have a wide range of uses, including analysis and testing of materials, and production of radioisotopes. Their capabilities are applied in many fields, within the nuclear industry as well as in fusion research, environmental science, advanced materials development, drug design and nuclear medicine.

The IAEA lists several categories of broadly classified research reactors. They include 60 critical assemblies (usually zero power), 23 test reactors, 37 training facilities, two prototypes and even one producing electricity; but most (160) are largely for research, although some may also produce radioisotopes. As expensive scientific facilities, they tend to be multi-purpose, and many have been operating for more than 30 years.

Today, Russia has most research reactors (62), followed by USA (54), Japan (18), France (15), Germany (14) and China (13). Many small and developing countries also have research reactors, including Bangladesh, Algeria, Colombia, Ghana, Jamaica, Libya, Egypt, Thailand and Vietnam. About 20 more reactors are planned or under construction, and 361 have been shut down or decommissioned, about half of these in USA. Many research reactors were built in the 1960s and 1970s. The peak number operating was in 1975, with 373 in 55 countries.

1.3 Uses of research reactors

1-Neutron beams are uniquely suited to studying the structure and dynamics of materials at the atomic level. Neutron scattering is used to examine samples under different conditions such as variations in vacuum pressure, high temperature, low temperature and magnetic field, essentially under real-world conditions.

2-Using neutron activation analysis, it is possible to measure minute quantities of an element. Atoms in a sample are made radioactive by exposure to neutrons
in a reactor. The characteristic radiation each element emits can then be detected.

3-Neutron activation is also used to produce the radioisotopes, widely used in industry and medicine, by bombarding particular elements with neutrons so that the target nucleus has a neutron added. For example, yttrium-90 microspheres to treat liver cancer are produced by bombarding yttrium-89 with neutrons.

4-The most widely used isotope in nuclear medicine is technetium-99m, a decay product of molybdenum-99*. It is produced by irradiating a target of U-235 foil with neutrons (for a week or so) and then separating the molybdenum-99 from the other fission products in a hot cell - tc Mo-99 being about 6% of the fission products. Most Mo-99/Tc-99 production has been using HEU targets, but increasingly LEU is favored and HEU is being phased out.

5-Technetium generators, a lead pot enclosing a glass tube containing the radioisotope, are supplied to hospitals from the nuclear reactor where the isotopes are made. They contain molybdenum-99, with a half-life of 66 hours, which progressively decays to technetium-99m, with a half-life of 6 hours. The Tc-99 is washed out of the lead pot by saline solution when it is required. It is then attached to a particular protein for administering to the patient. After two weeks or less the generator is returned for recharging, since it loses 22% of its product every 24 hours.

6-Research reactors can also be used for industrial processing. Neutron Transmutation Doping (NTD), changes the properties of silicon, making it highly conductive of electricity. Large, single crystals of silicon shaped into ingots, are irradiated inside a reactor reflector vessel. Here the neutrons change one atom of silicon in every billion to phosphorus. The irradiated silicon is sliced into chips and used for a wide variety of advanced computer applications. NTD increases the efficiency of the silicon in conducting electricity, an essential characteristic for the electronics industry.
In test reactors, materials are also subject to intense neutron irradiation to study changes. For instance, some steels become brittle, and alloys which resist embrittlement must be used in nuclear reactors.

Like power reactors, research reactors are covered by IAEA safety inspections and safeguards, because of their potential for making nuclear weapons. India's 1974 explosion was the result of plutonium production in a large, but internationally unsupervised, research reactor which closed at the end of 2010.

One of the more interesting and powerful test reactors was Plum Brook in Ohio, USA, which operated for NASA over 1961-1973 and was designed to research nuclear power for aircraft, then nuclear-powered rockets and spacecraft. It was 60MW pool-types, light water cooled and moderated, with a very high neutron flux 420 trillion/cm²/sec.

Our study deals with Egyptian Testing and Research Reactor no. 2 (ETRR-2) which is one of low enriched Uranium MTR reactor type M. A. Raouf et. al[3]. The Egypt’s second research reactor is a pool-type with an open water surface and variable core arrangement. The core nominal power was planned to be 22MW, cooled by light water, moderated by water and beryllium reflectors.

The design parameters are outlined in table [1.1], M. A. Raouf et. al[3]

Table [1.1] ETRR-2 design parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design thermal power, MW</td>
<td>22</td>
</tr>
<tr>
<td>Water temperature at core inlet, °C</td>
<td>40</td>
</tr>
<tr>
<td>Water temperature at core outlet, °C</td>
<td>50</td>
</tr>
<tr>
<td>Nominal core flow, m³/h</td>
<td>1900</td>
</tr>
<tr>
<td>Number of fuel elements</td>
<td>29</td>
</tr>
<tr>
<td>Number of fuel plates per fuel element</td>
<td>19</td>
</tr>
<tr>
<td>Enrichment, %</td>
<td>19.7</td>
</tr>
<tr>
<td>Total peaking factor</td>
<td>3.0</td>
</tr>
<tr>
<td>Axial peaking factor</td>
<td>1.35</td>
</tr>
<tr>
<td>Parameter</td>
<td>Value</td>
</tr>
<tr>
<td>-----------------------------------------------------</td>
<td>----------------</td>
</tr>
<tr>
<td>Active length, cm</td>
<td>80.0</td>
</tr>
<tr>
<td>Extrapolated length, cm</td>
<td>8.0</td>
</tr>
<tr>
<td>External section of fuel element, cm^2</td>
<td>8.0 x 8.0</td>
</tr>
<tr>
<td>Section in grid to house the FE, cm^2</td>
<td>8.1 x 8.1</td>
</tr>
<tr>
<td>Plate thickness, cm</td>
<td>0.15</td>
</tr>
<tr>
<td>Fuel thickness, cm</td>
<td>0.07</td>
</tr>
<tr>
<td>Fuel width, cm</td>
<td>6.40</td>
</tr>
<tr>
<td>Frame thickness, cm</td>
<td>0.50</td>
</tr>
<tr>
<td>Frame width, cm</td>
<td>8.0</td>
</tr>
<tr>
<td>External distance between frames, cm</td>
<td>8.0</td>
</tr>
<tr>
<td>Internal distance between frames, cm</td>
<td>7.0</td>
</tr>
<tr>
<td>Water gap between plates:</td>
<td></td>
</tr>
<tr>
<td>of a single fuel element, cm</td>
<td>0.27</td>
</tr>
<tr>
<td>of different fuel elements, cm</td>
<td>0.39</td>
</tr>
<tr>
<td>Upper plenum, cm</td>
<td>12.0</td>
</tr>
<tr>
<td>Inlet channel length, cm</td>
<td>15.0</td>
</tr>
<tr>
<td>Inlet channel external section, cm^2</td>
<td>8.0 x 8.0</td>
</tr>
</tbody>
</table>

The core is built on a supporting grid having 6 x 5 positions available for placing fuel or irradiation boxes. Figure (1.1) shows the basic core arrangement M. A. Raouf et. al[3]. The core grid is wrapped by double Zircalloy square sheets, leaving an empty volume between the sheets. The empty volumes corresponding to each reactor face are independent. The core is cooled by light water flowing in an upwards forced flow. The neutron moderator is light water, while the reflectors are Beryllium and light water. The average neutron thermal flux of $1.3 \times 10^{14}$ n/(cm$^2$ sec) in the central flux trap. The neutronics feedback coefficients are negative. The core is optimized for medical quality Cobalt production of $1.85 \times 10^{15}$ Bq/yr (50000 Ci/yr) of Co$^{60}$
with a specific activity $7.4 \times 10^{12}$ Bq/gr. (200 Ci/gr.). The core is made of square shaped fuel elements, with an active length of 0.8 m. Each fuel element is composed of 19 plates with a 19.7% enriched Uranium load as shown in Figure (1.2), the clad is Aluminum. A coolant channel is built between adjacent fuel plates to enable removal of the energy delivered in the fission reaction.

**Fig. (1.1) Basic core arrangement.**

FE = Fuel element
Fig. (1.2) Fuel plate assembly.
The core control is accomplished through 6 Ag-In-Cd alloy absorbing plates. The plates are fully withdrawn position is at the core's top; their hydraulic driving mechanisms are located in a premise below the reactor pool. The plates are placed inside fixed channels that run across the core, the Natural Convection Core Cooling system (NCCC) will enable, after a reactor shutdown or during low power operation, to cool the core by natural water circulation.

The main primary system shielding is built of heavy concrete, and its thickness is such that the operational exposure doses of the operating staff are below prescribed limits [4]. The Reactor Protection System (RPS) commands two diverse and independent reactor shutdown systems that extinguish the nuclear reaction. After reactor shutdown decay heat is removed by natural circulation flow mechanism through flap valves.

1.4 Cooling description

This type of cooling is achieved by the opening of two Flap Valves (FV), each one of them located on the primary coolant return pipes inside the reactor pool as shown in Figure (1.3); the flap valves are held closed by the action of the primary cooling system flow in its design direction. The weight of the valve disk is counter balanced by the action of the forces of pressure on the disk faces. Immediately after turning off the primary pumps, inertia flywheels (one per each pump) will maintain during a certain period of time the Forced Core Cooling Convection (FCCC). When pressure in the circuit has descended sufficiently or if the flow reverses, flap valves will open by gravity and natural core cooling circulation will start. Hot water from the core will rise through the chimney while cold water from the pool will come down through the flap valves and the primary return piping towards the core. This cooling mode of the core is called the Natural Convection Core Cooling system (NCCC).
It is estimated that this core cooling mode can also be achieved, though degraded, with only one of the flap valves open. Consequently, it shall be considered that the NCCC will fail when both flap valves fail to open up on demand. It is interesting to note that both flap valves will also act as Siphon Effect Breakers (SEB) in case of primary Loss of Coolant Accident (LOCA), due to ruptures in the primary circuit on the return side to the reactor tank.

On the other hand, when confronted with an uncontrolled coolant loss from the reactor pool, the NCCC will remain for only as long as the pool water level may take to reach the flap valves.

1.4.1 Cooling circuits

Multi-purpose reactor core cooling system divided into two independent circuits with one pump and one heat exchanger each.

1. Primary cooling circuit: consists of several systems, these systems are:
   a- Core cooling system
   b- Pool cooling system, and
   c- Auxiliary pool cooling system.

2. Secondary cooling circuit (cooling tower).

1.4.2 Core cooling system

The core described by ETRR-2 is cooled by forced upward flow as shown in Figure (1.4) and Figure (1.6), after the water leaves the core it is driven through a lateral pipe connecting to the chimney. Outside the reactor pool boundaries the primary coolant flow is split in two 50% capacity branches as shown in Figure (1-6), each branch has two 50% centrifugal pumps and one plate-type heat exchanger.

The system has two normal operation modes: two branches working with one pump operating in each branch, and a single branch working with a single pump for powers lower than 11MW. With the two branches in operation the following
variables are supervised: the flow rate for each branch, the inlet and outlet temperatures of each heat exchanger, the water conductivity, the pumps pressures and the differential pressure at the core. For single branch operation the valves state is the same as for two branches operation. The branches independence is maintained by non-return valves which prevent the flows from mixing.

1.4.3 Pool cooling system

The reactor pool cooling system is cooled by a dedicated system as shown in Figure (1.5). The coolant flows in a downward direction and after leaving the reactor pool it goes through a N16 decay tank. Two 100% centrifugal pumps and a plate type heat exchanger are provided. The maximum cooling capacity is approximately 1.5MW and the nominal flow rate 185 m³/hr,[5].

![Fig. (1.3) Natural circulation flow path.](image)
Fig. (1.4) Core thermal hydraulic.

Fig. (1.5) Pool cooling system.
Fig. (1.6) Reactor core cooling system.

Fig. (1.7) Secondary cooling system.
1.4.4 Secondary cooling system

The secondary cooling system in Figure (1.7) removes the heat from the primary system heat exchangers and other systems heat exchangers and transfers it to the environment through cooling tower. The cooling towers are induced draft type and they are arranged in six cells, each cell has its own blower, the pool for the six cells, has a water volume of 400 m$^3$/hr approximately; [6].

During the stop periods, the reactor uses two 35 m$^3$/hr centrifugal pumps to control the reactor pool and auxiliary pool temperatures. Given the wet-bulb temperature of 24°C, for an occurrence of the 2.5% during the four warmest months according to ASHREA for Cairo and conservative approximation of 6°C the outlet cold water is 30°C and the given flow rate is 3120 m$^3$/hr. and the thermal load of 25MW, the cooling range is approximately 7°C, therefore the towers inlet hot water temperature is 37°C.

1.5 Heat exchanger in MTR

The heat exchangers used are of Alfa-Laval plate type consisting of 162 of corrugated plates. In the heat exchanger the primary coolant flows upward on one side of the plates and the secondary coolant flows downward on the other side. The secondary coolant system removes the heat from the primary system heat exchangers and transfers it to the environment through cooling tower units. A plate heat exchanger is employed in the reactor core and pool cooling system. This plate heat exchanger as shown in Figure (1-8) is designed according to ASME VIII DIV. 1 (American Society of Mechanical Engineering), the thermal-hydraulic characteristics of the core heat exchanger is illustrated below in table [1.2], H. H. El Khatib [7]
Fig. (1.8) Exploded View of Plate Type Heat Exchange.

Table [1.2] Plate heat exchanger specifications, H. H. El Khatib [7]

<table>
<thead>
<tr>
<th>Side</th>
<th>Hot stream</th>
<th>Cold Stream</th>
</tr>
</thead>
<tbody>
<tr>
<td>Medium</td>
<td>Water</td>
<td>Water</td>
</tr>
<tr>
<td>Flow rate (kg/h.)</td>
<td>1000000</td>
<td>1350000</td>
</tr>
<tr>
<td>Temperature</td>
<td>49.5 °C to 40 °C</td>
<td>30 °C to 37 °C</td>
</tr>
<tr>
<td>Pressure drop</td>
<td>76 KPa</td>
<td>120 KPa</td>
</tr>
<tr>
<td>Heat transfer area</td>
<td>292.56 m²</td>
<td>292.56 m²</td>
</tr>
</tbody>
</table>
1.6 Cooling tower in MTR

Fig. (1.9) cooling tower cell.

Cooling tower in ETRR-2 is used to extract generated heat in the reactor core and dissipates it to atmosphere as shown in Figure (1.7). The configuration of cooling tower is illustrated in Figure (1.9) is Induced draft mixed flow wet cooling tower, the thermal-hydraulic characteristics of the tower is shown in the table [1.3]; H. H. El Khatib [7]

Table [1.3] Cooling tower specifications:

<table>
<thead>
<tr>
<th>Type</th>
<th>Induced draft cross flow</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tower water flow.</td>
<td>3124 m³/hr</td>
</tr>
<tr>
<td>Tower capacity.</td>
<td>2,548,000,0 kcal/h.</td>
</tr>
<tr>
<td>Hot water temperature.</td>
<td>37.0°C</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>------------------</td>
<td>-------------------</td>
</tr>
<tr>
<td>Range.</td>
<td>7.0°C</td>
</tr>
<tr>
<td>Cold water temp.</td>
<td>30.0°C</td>
</tr>
<tr>
<td>Approach.</td>
<td>6.0 °C</td>
</tr>
<tr>
<td>Wet-Bulb temp.</td>
<td>24.0°C</td>
</tr>
<tr>
<td>Dry-Bulb temp.</td>
<td>37.41°C</td>
</tr>
<tr>
<td>Relative humidity</td>
<td>50%</td>
</tr>
<tr>
<td>Humidity ratio out.</td>
<td>0.0318</td>
</tr>
<tr>
<td>Flow fan tower inlet.</td>
<td>198400 cfm</td>
</tr>
<tr>
<td>Thermal analysis.</td>
<td></td>
</tr>
<tr>
<td>Water rate.</td>
<td>22.32 gpm/ft²</td>
</tr>
<tr>
<td>Dry air rate</td>
<td>51.04 lb/min/ft²</td>
</tr>
<tr>
<td>Liquid to air ratio (L/G).</td>
<td>1.366</td>
</tr>
<tr>
<td>Tower coefficient. (KaV/L).</td>
<td>1.046</td>
</tr>
</tbody>
</table>

### 1.7 Postulated initiating events (PIE) and accident scenarios potentially leading to core damage in research reactors

Initiating events originate from component failures, system malfunctions, human errors, external events or a combination of these, either in the reactor itself or in one of its experimental devices. Postulated initiating events (PIEs) should be grouped according to both their impact on the integrity of the reactor core or other components and the protective actions designed to deal with an occurrence of the events. The main reason for grouping PIEs is to analyze quantitatively only the limiting cases of each group. The following is an appropriate grouping or categorization that takes into account the categorization used for PIEs recommended.

i- Loss of electrical power supplies;

ii- Insertion of excess reactivity;
iii- Loss of flow;
iv-Loss of coolant,
v- Erroneous handling or malfunction of equipment or components;
vi-Special internal events;
vii- External events.

It should be emphasized that not every group of PIEs is applicable to every type of reactor. On the contrary, some of the above groups may not be applicable to many types of research reactor; for example, excess reactivity insertions (group 2) may not be applicable to reactors having very large negative temperature coefficients of reactivity, and unbalanced heat removal (groups 3 and 4) may not be applicable to low power reactors. Another major cause of system or component malfunction is the handling or improper re-installation of systems or components during maintenance or repair work. This is not explicitly mentioned among the PIEs, since it may be the cause of any of the PIEs listed. A more detailed discussion of the most important PIEs is given in the subsections below.

1.7.1 Power excursion due to insertion of excess reactivity.

Insertion of excess reactivity can lead to a source term because of its potential to deposit significant amounts of thermal energy into the fuel at a rapid rate. This initiating event is postulated to determine a limiting specific insertion of reactivity, covering accidents that may happen because of failures during:

i- Critical experiment;
ii- Reactor startup;
iii-Fuel element loading or unloading;
iv- Manipulation or operation of equipment close to the reactor core or Other components including experimental devices; or
v- Removal of large absorbers from the core.

This type of PIE is usually analyzed in the SAR of a research reactor because of the frequent changes of core configurations or experimental devices that take place during the reactor lifetime.

The accident scenarios associated with this PIE usually lead to the meltdown of a fraction of the reactor core if no credit is given to the actuation of the protection or shutdown systems. The meltdown fraction may range from one or a few fuel plates or rods to a significant fraction of the core, depending on the characteristics of the reactor and accident sequence. These accidents may be analyzed with system codes such as PARET and RELAP5, which include different degrees of complexity.

The amount of fuel that may melt can be estimated on the basis of experimental research or through conservative simple manual calculations. However, on certain occasions, more sophisticated models and codes are used to estimate the source term, particularly for higher power research reactors. Among these are codes used in the source term analysis of severe accidents in power reactors.

In any cases, a precise estimate of the fraction of the fuel meltdown requires thermo-hydraulic calculations, which tend to be very sensitive to input parameters. In particular, some specific fuel element features, such emissivity of the fuel plate surface, will greatly influence the temperature distribution and the maximum fuel temperature of the reactor after the excursion.

1.7.2 Loss of flow accident (LOFA.)

Reduction of the coolant flow as a result of various initiating events such as pump or valve failure, channel blockage or flow redistribution may lead to cladding failure due to overheating. The probability of such an accident depends on the design of the reactor core and the type of forced flow (upward or downward). However, most SARs of research reactors include one or more of these accidents leading to fuel failure and the need for source term evaluation.
For some research reactors, these accidents are considered as Design Basis Accidents (DBAs), and some Engineering Safety Factors (ESFs) are built in to cope with their consequences.

Loss of flow accidents under full power or decay heat conditions can lead to a significant reduction of heat removal capability. Loss of flow condition can result from several causes, including
- Loss of pumping power;
- Core inlet flow blockage;
- Loss of pressure in the cooling system.

A principal mechanism contributing to damage propagation is the onset of flow excursions induced by parallel channel (i.e. Ledinegg type) instability. This can be initiated by the onset of significant voiding (in the sub-cooled boiling flow regime) in any of the parallel flow channels. The well-known Saha–Zuber correlation; S. G. Kandlikar [8]; that predicts the point of net vapor generation may be used for this purpose. Voiding can be initiated in flow channels as a result of system depressurization, flow blockages from debris or manufacturing defects, or loss of circuit flow. Flow starvation in several channels would require that the nuclear heat be transferred to the coolant in neighboring unaffected channels. If the heat generated in affected channels cannot be dissipated, the fuel heat-up to melting may occur. Upon melting, fuel foaming (with the amount being dependent on burn-up) can lead to plate contact with neighboring plates, and hence to propagation of damage in the fuel assembly. Another important feature of many reactors is that liquid coolant flows downward through the core. This feature is undesirable from the view of damage initiation and propagation, especially for high power reactors. Any debris lodged at the entrance to the core will tend to stay entrapped. Also, during loss of pumping accidents, coolant flow reversal from forced to natural convection modes will lead to periods of stagnation and possibly to the onset of core damage.
The accident scenarios range from the damage of one or several fuel plates (cladding failure) due to overheating to the melt down of one or several fuel plates or even fuel elements in research reactors using MTR type fuel. The event sequence depends on the specific scenario. This reduced flow condition eventually undergoes a transition to natural convection cooling, in the case of pump coast down, or to stagnant coolant followed by rapid coolant voiding (e.g. steam explosion), in the case of core blockage.

1.7.3 Loss of coolant accident (LOCA)

The loss of coolant accident (LOCA) may lead to fuel damage resulting in a source term in the core of a research reactor above a certain power level. This level varies according to the fuel design and is typically around 2MW for research reactors using plate type fuel [8]. Higher power reactors usually incorporate appropriate ESFs to avoid such an accident. However, the consequences of a LOCA need to be examined for all reactors, including those at the lower end of the range, to determine the likelihood of fuel damage. Even if the likelihood is found to be very low, the potential for direct irradiation of operating staff or personnel due to loss of shielding must also be examined.

A LOCA leading to partial uncovering of the core should be analyzed to determine its potential for fuel damage. In some cases, it could result in boiling of the water adjacent to the immersed part of the fuel. Although in this case conduction and steam cooling may be adequate to prevent melting, a cladding failure could take place.

Most research reactors with power levels above 2MW have incorporated adequate ESFs such as anti-siphon devices, elevation of primary pool pipe work above the reactor core, use of pool liner throughout beam tubes, automatic operation of the beam tube shutter and redundant water storage tanks plus core spray systems or other emergency core cooling systems (ECCSs). However, the SARs of some of these reactors include small LOCAs with the meltdown of a
fraction of the core as a DBA. In these cases, it is advisable to conduct a careful study of the most limiting LOCA sequence that could lead to fuel damage. This study should take into account the action (or failure) of the incorporated ESFs or of any other components that are vital to ensuring core cooling. The frequency of this accident sequence should be shown to be low enough to demonstrate that the overall risk associated with the meltdown of large portions of the core is acceptable.

Although parallels may be drawn between LOCAs and LOFAs, flow degradation under LOCA conditions is assumed to occur when the water level in the reactor core has dropped significantly (e.g. due to a ruptured beam tube), such that fuel plates can transfer energy only to a gaseous environment. Under these conditions, if a sufficient heat sink does not exist, then core melting and subsequent melting propagation could occur, starting from the regions of highest power densities. Once again, fuel foaming needs to be considered under high burn up conditions. Fuel melting and relocation downward can be expected to occur in a candling type mode, or as rivulets. One aspect to be concerned about is associated with the formation of a molten aluminum pool at the lower core support plate.

1.7.4 Fuel handling accidents

Fuel handling accidents include fuel dropping, dropping of the transfer cask on fuel elements and fuel uncovering. The potential hazards are mechanical damage to the fuel, insufficient cooling leading to melting and the possibility of a reactivity accident.

A fuel handling accident in which a fuel element is dropped could lead to mechanical damage to the fuel or heat-up of the fuel element to melting if for example, channel flow reduction occurs. The release of fission products in this case, however, is limited and comparable with that resulting from a fuel channel blockage. If the fuel element has an extensive cooling period, no melting of fuel
plates will occur; however, for accident analysis and source term derivation, limiting conditions are to be assumed and no credit is to be taken for the cooling period. For metallic fuels used in research reactors, fission products will remain trapped within the fuel as long as the fuel temperature remains below the level at which blistering occurs? It is important that possible spent fuel melting or failure also be included in the source term evaluation for fuel handling events in the spent fuel bay. This is particularly important for spent fuel that is stored within the containment structure. For separate spent fuel storage facilities, the possibility of a reactivity accident due to organizational errors or erroneous handling of spent fuel may need to be assessed as possible input for a source term evaluation.

1.7.5 Flow redistribution condition

Research reactor operating conditions are defined by power, pressure, coolant inlet temperature, coolant flow rate and pressure drop through the core, variables that are common to all parallel channels. The operating point of each channel is fixed by the intersection of two curves:

a- Pressure drop through the channel as a function of flow rate
b- Core pressure drop, which is constant.

This channel characteristic according to the imposed flow conditions, coolant inlet temperature and pressure, has two inflection points. This leads to non-unique operating points for channels for some conditions, allowing the potential to move from one to the other, if condition are such that the flow rate decreases dramatically, with consequent significant increase in vapor bubble formation, the effect is known as Ledinegg excursion or flow redistribution.
1.8 Power peaking factor related design basis

The total Power Peaking Factor (PPF) is related to the non-homogeneous spatial distribution of the heat flux over the core. This distribution determines that in certain spots of the core the local heat flux is higher than the average value, for thermal-hydraulic calculations a cosine profile with a maximum PPF value of 3 is conservatively adopted.

For every core operational configuration, it must be fulfilled that PPF≤3, Wm. J. Garland et al [20].

The average and the maximum heat fluxes are calculated in the following way:

a-The average heat flux is calculated as the power removed by primary cooling system divided by the total heat transfer area.

b-The maximum operational heat flux is obtained from the product between the average heat flux and the PPF.

1.9 Safety relevant design basis

Research reactors that are cooled and moderated by water are limited, from the thermal point of view, by critical phenomena leading to boiling crisis and damage in the fuel assemblies.

Power is established to a maximum within reasonable safety margins compatible with the imposed operative conditions. To establish those margins it is necessary to know the limits above which critical phenomena appear leading to a rapid increase in cladding temperature and fuel plate damage.

1.9.1 Critical phenomena

Different phenomena can lead to the boiling crisis and it considered of importance to define the design criteria. These phenomena relevant to the reactor safety are:

1-Departure from Nucleate Boiling (DNB)
2-Flow instability phenomenon
3-Low flow burn-out phenomenon.
4-Fluid-structure interaction (critical velocity)

1.9.1.1 Departure from nucleate boiling.

The Onset of Nucleate Boiling (ONB) is considered as the first indication of the potential for critical phenomena, and the heat flux that indicates ONB is frequently used as a thermal design constraint. The ONB is taken as a warning in steady state conditions as it doesn’t actually correspond to any critical event. For reactor design purposes, an acceptable prediction method for burnout is needed since departure from nucleate boiling is potentially a limiting design constraint. Burnout occurs due to vapor blanketing on the heated wall, and the resulting Critical Heat Flux (CHF) depends only on the local conditions considering that the effect of flow orientation up-flow or down-flow is small. This burnout mechanism is a DNB type. The heat flux leading to this situation is named CHF, burn-out flux or departure from nucleate boiling (DNB) flux.

1.9.1.2 Flow instability phenomena.

The most common flow instabilities encountered in heated channels with forced convection are the flow excursion and density wave oscillation types. The flow excursion or Ledinegg instability is initiated (but, in some transient conditions, may not be completed) when the slope of the channel demand pressure drop-flow rate curve become algebraically smaller than or equal to the slope of the loop supply pressure drop-flow rate curve. The channel pressure drop-flow curve depends on the channel geometry, inlet and exit resistances, flow direction, sub-cooled steam void fraction and heat flux, along the channel. There is a critical value for the inlet sub-cooling below which flow instability would be caused by density wave oscillations only and no flow excursion would occur.
For most research reactors, the steady state operating system pressure is low and the inlet coolant temperature is much lower than the saturation temperature. It can, therefore, be concluded that a flow excursion would occur for a given flow rate at high enough heat flux, or density wave oscillations will not occur under normal operating conditions. For these reasons this phenomena is limited in research reactors but this phenomena could lead to cladding failure and it may occur as a result of:

a- A temporary power increase
b- A flow rate decrease such as, a loss of flow in primary coolant circuit.

1.9.1.3 Low flow burn-out

A periodic instability known as pulsed boiling can appears for low power, low coolant velocities. The sequence of events characterizing this phenomenon is as follows:

a- Convection and onset of nucleate boiling, the coolant inside the channel, where the maximum axial heat flux takes place, begins to heat up, bubbles appears and pressure increases.
b- Coolant expulsion, vapor flashes at both ends of the coolant channels.
c- Re-introduction, sub-saturated coolant re-enters the channel and the sequence starts again.

When the heat flux is approximately 2 to 4 times greater than the pulsed boiling heat flux, burnout appears. There is a large and permanent increase in the wall temperature due to a vapor film formed at the hot spot, leading to fuel damage.

1.9.1.4 Fluid-structure interactions

When a high flow passes through a gradually narrowing passage, the pressure head of the stream is converted into a velocity head, which creates a suction force on the wall. If the wall is movable or flexible, the flow area could
be reduced to zero and the flow stopped. As soon as the flow becomes stagnant, the stream pressure increases, pushing the wall back, thus providing a wide flow area again. These suction and pushing forces can act periodically and makes the structure vibrate. This mechanism appears to govern the vibration of parallel fuel plates. These hydraulic vibrations can result in a large deflection of the plates, causing local overheating and possibly a complete blockage of the coolant channels.

1.10 Non-safety relevant design basis

In research reactors using plate–type fuel assemblies, specific thermal-hydraulic conditions are commonly used as design constraints or warning in order to anticipate critical phenomena appears before the critical phenomena are referred as non-safety relevant condition. The only non-safety relevant condition for this design in the onset of nucleate boiling (ONB), in this sense a design basis is adopted for the cooling system requiring that it be designed in such a way that it provides enough cooling to the reactor core in any operational situation, and local boiling should not be reached in any point during steady state normal operation.

1.11 Objectives

The accident conditions are a main problem in research reactor, and the proposed model will focus on the analysis of steady and transient states of Egyptian testing and research reactor no.2, the model is formulated to simulate the cooling systems in both cases in regime I and regime II, so it is used to simulate the loss of ultimate heat sink presented by a failure of six cell unit of cooling tower one by one till complete failure of the cooling tower to predict the behavior of reactor protection system. It is a time dependent problem which estimates the triggering time of safety system depending on the available heat sink, also the different heat transfer margins will be estimated like Onset of
Nucleate Boiling (ONB), Onset of Flow Instability (OFI) and departure from nucleate boiling (DNB) to evaluate the safety settings equipped and stored in the Reactor Protection System (RPS).
CHAPTER (2)

LITERATURE REVIEW

In recent years, a general interest in the evaluation of the research reactor performance and operational characteristics by the International Atomic Energy Agency (IAEA) has been generated. This interest is directed towards developing simulation programs for PC use and is concentrated on thermal-hydraulic calculations for research reactors transients. The analysis of the transient behavior of research reactors has received great attention since a lot of time because of its importance in determining the limits of clad melting temperature. Generally the use of large codes requires a considerable amount of efforts and skills with respect, in particular, to input preparation and output processing. Sometimes the large codes cannot offer some details, which the reactor operator needs to know, about the initiating events’ behavior during the proposed accident scenarios like reactivity insertion due to control rod withdrawal; this withdrawal may be ended or fixed (stuck), or may be continued during the over power scram trip. Indeed, contrary to power reactors, research reactor operation is characterized by frequent core modifications, as a result of the change in experimental needs. Practically, each core modification must satisfy a number of safety criteria. Hence, it is desirable for the operator to dispose means to perform simple and realistic transient estimations, even if only for scooping purposes.

The survey is divided into three categories, the first is related to the critical phenomena that limit the research reactor power, the second is the thermal hydraulic analysis and simulation of research reactors and the last is related to the simulation of the heat exchangers and cooling towers.
2.1 Critical phenomena limiting research reactor power

Prediction of the onset of the flow instability (OFI) in steady and transient sub-cooled flow boiling is an important consideration in the design and operation of nuclear reactors, in particular for materials testing reactors (MTR). Khater et al [10] developed a predictive model for OFI in the MTR. The model was based on both the heat balance during the bubble generation and condensation processes, and the force balance for the detached bubbles at the onset of significant void (OSV). The only adjustable coefficient involved in the proposed model was quantified by comparison with the experimental data of Whittle and Forgan, which covers the wide range of MTR operating conditions. The model predictions were compared with predictions of some previous models, and it was shown that the model results in smaller deviation from the experimental data. A correlation for the heat flux at OFI was also developed based on this model. The correlation gave lower deviation from the experimental data than the well-known correlation of Whittle and Forgan [64]. The model was also used to predict the OFI locus during a transient, where it showed good agreement with the short transient data Prediction of the onset of flow instability in transient sub-cooled flow boiling.

The predictive model of the onset of flow instability, OFI developed and validated previously was applied by Khater et al [11] on the Egyptian second research reactor (ETRR-2) under loss of flow transient. Both best-estimate and conservative calculations were performed at the hot channel for both exponential and ramp pressure gradient change. The OFI locus was depicted and plotted against the coolant velocity, the exit coolant temperature and the bubble detachment parameter for several heat fluxes. It was found that, OFI occurred at exit coolant temperature value greater than about 105°C for the best estimate calculation and about 107°C for the conservative calculation where the heat fluxes leading to OFI phenomenon occurred at a bubble detachment
parameter value lower than about 30 for all calculations. The safety margins for OFI were also predicted where their values in the best estimate calculation were 2.62.

A new empirical correlation to predict the sub-cooling at the onset of flow instability in vertical narrow rectangular channels was developed by El-Morshedy \[12\]. The developed correlation involves almost all parameters affecting the phenomenon in a dimensionless form and the coefficients involved in the correlation were identified by the experimental data of Whittle and Forgan that covers the wide range of MTR operating conditions. The developed correlation gave a much lower relative standard deviation of only 6.6% from the experimental data of Whittle and Forgan. The bubble detachment parameter was also estimated based on the present correlation. The predictive correlation was then utilized in a model predicting the void fraction and pressure drop in sub-cooled boiling under low pressure and then used to predict the S-curves representing the two phase instability with good accuracy. The correlation was also incorporated in the safety analysis of the IAEA 10MW MTR generic reactor in order to predict the OFI phenomenon under both fast and slow loss-of-flow transient. The OFI locus for the reactor coolant channels was predicted against flow velocity, exit temperature and bubble detachment parameter for various heat flux values. It was found that the reactor had vast safety margins for OFI phenomenon under both steady and transient states.

In research reactors, plate-type fuel elements were generally adopted so as to produce high power densities and are cooled by a downward flow. A core flow reversal from a steady-state forced downward flow to an upward flow due to natural convection was occurred during operational transients such as "Loss of the primary coolant flow". Therefore, in the thermal hydraulic design of research reactors, critical heat flux (CHF) under a counter-current flow
limitation (CCFL) or a flooding condition were important to determine safety margins of fuel against CHF during a core flow reversal. Kaminagat, et al [13] had proposed a CHF correlation scheme for the thermal hydraulic design of research reactors, based on CHF experiments for both upward and downward flows including CCFL condition. When the CHF correlation scheme was proposed, a sub-cooling effect for CHF correlation under CCFL condition had not been considered because of a conservative evaluation and a lack of enough CHF data to determine the sub-cooling effect on CHF. A too conservative evaluation was not appropriate for the design of research reactors because of construction costs etc. Also, conservativeness of the design was determined precisely. In this study, therefore, the sub cooling effect on CHF under the CCFL conditions in vertical rectangular channels heated from both sides were investigated quantitatively based on CHF experimental results obtained under uniform and non-uniform heat flux conditions. As a result, it was made clear that CHF in this region increased linearly with an increase of the channel inlet sub-cooling and a new CHF correlation including the effect of channel inlet sub-cooling was proposed. The new correlation could be adopted under the conditions of the atmospheric pressure, the inlet sub-cooling less than 78K, the channel gap size between 2.25 to 5.0 mm, the axial peaking factor between 1.0 to 1.6 and L/De between 71 to 174 which were the ranges investigated.

A correlation for the counter-current flow limitation (CCFL) in vertical rectangular channels was applied by Sudo, et al [14] to predict the critical heat flux (CHF) for downward flow in a vertical rectangular channel heated from both sides. The new correlation was based on CCFL experiments carried out with a two-phase flow system of air and water under 1 atm. with a superficial air velocity range of 1–17 m/s. The CHF experiments were carried out with an inlet water sub-cooling range of 25–75°K and an inlet water mass flux range of 2–600 kg/m²s under about 1 atm. for a 750 mm long, 50 mm wide and 2.25 mm
gap flow channel and a 375 mm long, 50 mm wide and 2.80 mm gap flow channel which were heated from both sides. The comparison between the prediction based on the new correlation for the CCFL and the CHF experimental results could provide a good quantitative understanding of CHF characteristics, which was required for the thermal-hydraulic design and safety analysis of nuclear research reactors in which the downward flow was adopted for core cooling with flat-plate-type fuel. The role of the aspect ratio of the rectangular channel was evident in both the CCFL and CHF characteristics for countercurrent flow and it was strongly implied that the CHF for downward flow was at a minimum under the flooding condition in the case of large inlet water subcooling and when the inlet downward water mass flux was greater than that under the flooding condition in the case of small inlet water subcooling.

The effect of heated length on critical heat flux (CHF) in thin rectangular channels under atmospheric pressure had been studied by Tanaka et al [15]. CHF in small channels had been widely studied in the last decades but most of the studies were based on flow in round tubes and number of studies focused on rectangular channels is relatively small. Although basic triggering mechanisms, which leaded to CHF in thin rectangular channels, are similar to that of tubes, applicability of thermal hydraulic correlations developed for tubes to rectangular channels were questionable since heat transfer in rectangular channels were affected by the existence of non-heated walls and the noncircular geometry of channel circumference. Several studies of CHF in thin rectangular channels had been reported in relation to thermal hydraulic design of research reactors and neutron source targets and correlations had been proposed, but the studies mostly focused on geometrical conditions of the application of interest and therefore effect of channel parameters exceeding their interest was not fully understood. In this study, CHF data for thin rectangular channels had been collected from previous studies and the effect of heated length on CHF was
examined. Existing correlations were verified with data with positive quality outlet flow but none of the correlations successfully reproduced the data for a wide range of heated lengths. A new CHF correlation for quality region applicable to a wide range of heated lengths had been developed based on the collected data.

2.2 Thermal-hydraulic analysis and simulation in MTR

With the development of best estimate methods in nuclear safety analysis different codes have been applied for the simulation of transients to reduce the level of the uncertainties and to mitigate conservative assumptions in order to optimize design and safety features of nuclear reactors. In this respect, the thermal hydraulic system code RELAP5 was applied for the analysis of thermal hydraulic behavior of the German research reactor FRJ-2 by Di Maro [16] for the case an anticipated hypothetical reactivity accident. The analysis aimed at testing the capability and reliability of RELAP5 for research reactor analysis which had been mainly applied in the past for analysis of nuclear power plant. The results of the simulation had been compared with comprehensive calculation of the same type of transient using the verified two phase system code (CATHENA). In both simulations it was clear that the hypothetical accident considered would cause damage in the core. Under this accident condition the cooling system is not able to remove the heat released in the fuel elements. Due to forced convection only a small fraction of the external reactivity added to the core was compensated by the temperature feedback mechanism. In general, the results of the two codes were in agreement and comparable in the initial phase of the thermo-hydraulic processes taking place during the transient. After reaching the flow regime with fully developed nucleate boiling and two phase flow the RELAP5 code exhibits different prediction for the course of the transient and thermal hydraulic quantities
including void fraction, local flow rate and fuel temperature. The differences in the simulation were expected to be caused by the following:

- Heat transfer correlations
- Correlations and criteria for the transition boundaries between the flow regimes
- Probably applied numerical models.

Three dimensional CFD full simulations of the fast loss of flow accident (FLOFA) of the IAEA 10MW generic MTR research reactor are conducted by Salama [17]. In that system the flow is initially downward. The transient scenario starts when the pump coasts down exponentially with a time constant of 1 s. As a result the temperatures of the heating element, the clad, and the coolant rise. When the flow reaches 85% of its nominal value the control rod system scrams and the power drops sharply resulting in the temperatures of the different components to drop. As the coolant flow continues to drop, the decay heat causes the temperatures to increase at a slower rate in the beginning. When the flow becomes laminar, the rate of temperature increase becomes larger and when the pumps completely stop a flow inversion occurs because of natural convection. The temperature will continue to rise at even higher rates until natural convection is established, that is when the temperatures settle off. The interesting 3D patterns of the flow during the inversion process are shown and investigated. The temperature history is also reported and is compared with those estimated by one-dimensional codes. Generally, very good agreement is achieved which provides confidence in the modeling approach.

The natural convection heat transfer of water in narrow rectangular vertical channel was studied experimentally by El-Morshedy et al [18]. The water channel was represented by two adjacent stainless steel heating plates with a dimension of 800 mm active length, 70 mm in width and 2.7 mm in gap
thickness simulating a coolant channel of a typical material testing reactor under atmospheric pressure. Experiments were carried out under heat fluxes ranged from 2.7 KW/m² to 26.8 KW/m² to cover all possible heat fluxes in the single-phase liquid regime. Wall surface temperature, coolant bulk temperature and local Nusselt number profiles were experimentally investigated. The measured Nusselt number values were compared with the predicted values of both correlation for natural convection mode and correlation for combined natural and forced convection with 32.7% and 15.5% relative standard deviation. For both correlations respectively, a new correlation of the form:

$$\text{Nu} = \text{Gr}_d^{0.284} \text{Pr}^{0.12} (d_e/z)^{0.25}$$

was developed to be utilized in the thermal–hydraulics and safety analysis of the reactor. The developed correlation predicted the local Nusselt number distribution along the test section with 5.8% relative standard deviation from the experimental results.

A mathematical model was developed by El-Morshdy [19] to predict the closure flow in the ETRR-2 chimney. The model was validated by FEHT finite element program, where its prediction showed a very good agreement with the finite element program prediction. Then the model was used to produce a contour plots represents the flow maps in the core chimney for different closure flow. The flow maps showed that a part of the core flow starts to ascend to the pool at a closure flow less or equal to 0.16 m³/h. It was shown also that the stagnation depth corresponding to the nominal closure flow of 50 m³/h was about 1.0 m which was efficient to prevent the radioactive water from ascending to the pool and so minimizing the exposure rate at the pool surface. Based on the model results, it was also recommended to operate the reactor with a minimum closure flow of 10 m³/h where below this value the stagnation depth decreases sharply.
Three-dimensional simulation of the IAEA 10MW generic reactor under loss of flow transient was introduced by Salama et al [20] using the CFD code, Fluent. The IAEA reactor calculation is a safety-related benchmark problem for an idealized material testing reactor (MTR) pool type specified in order to compare calculation methods used in various research centers. The flow transients considered include fast loss of flow accidents (FLOFA) and slow loss of flow accidents (SLOFA) modeled with exponential flow decay and time constants of 1 and 25 sec. respectively. The transients were initiated from a power of 12MW with a flow trip point at 85% nominal flow and a 200ms time delay. The simulation showed comparable results as those published by other research groups. However, interesting 3D patterns were shown that are usually lost based on the one-dimensional simulations that other research groups have introduced. In addition, information about the maximum clad surface temperature, the maximum fuel element temperature as well as the location of hot spots in fuel channel was also reported. Generally, good agreement was obtained. Probably, the most important finding was that, possibly, two hot spots are generated during control plates scram, one, which spans larger area, at the center where the cladding is thinner, and one, which covers slanted area, at the corner of the cooling channel where the cladding is thicker.

Salama et al [21] studied a realistic simulation of the fully developed, single-phase, turbulent flow in the hot channel of a typical MTR reactor in which a three-dimensional thermal-hydraulic simulation has been performed. The hot channel was represented by two adjacent fuel plates with a dimension of 800 mm active length, 70 mm in width and 2.7 mm in gap thickness and is located in the hottest position of the core. Because of the existence of similarity planes, only quarter of the system is considered for CFD analysis representing quarter fuel meat, clad and coolant channel. Steady state normal operation
regime at the reactor nominal power with coolant flow velocity 4.7 m/s is considered. The results obtained in this work were verified by those obtained from a one-dimensional simulation code, MTRTHA. In order to compare with MTRTHA, modified variables were generated to match those obtained by the one-dimensional code for comparison, where a good agreement was achieved. It is also found that, the maximum clad surface temperature predicted by the 3D calculation located at the clad mid-width is higher than the ID prediction by about 8°C but still below the onset of sub-cooled boiling by adequate safety margin. The results showed the essential features of the hot channel through 3D patterns in both the flow field and the heat transfer; the fuel, clad and coolant temperature.

Salama et al [22] studied and reported a two-dimensional CFD simulation of both partial and total blockage of a coolant channel in a fuel assembly of the IAEA 10MW generic material testing reactor under steady-state normal operation. The obstructed channel was subjected to a partial blockage as a sequence of continuous area contractions due to buckling of its fuel-plates inwards probably, as a result of manufacturing imperfection, thermal elongation or hydraulic instability. The simulation was performed under both the hot and average channel condition, where in the hot channel simulation, blockages up to 90% were demonstrated until boiling was achieved and the simulation was then stopped as kinetic with feed-back which was not available in Fluent code became essential to obtain a realistic simulation. In the average channel, the simulation was performed for 95% and 100% blockage where the predicted temperatures were below boiling temperature. The velocity and temperature profiles through the obstructed channel and the other channels as well as the streamlines at the entrance were presented and discussed for each blockage case. The CFD analysis provided an in-depth investigation to the interplaying thermal–hydraulic processes involved in this system that were usually missed
when using lumped analysis 1D codes. It was found that, the coolant flow was redistributed among the model channels according to the new hydraulic resistance in each. The obstructed (left) channel received less flow and the adjacent (middle) channel more flow while the flow in the third (right) channel remained unchanged. That leaded to new temperature distributions through the model fuel plates and coolant channels.

The 2D, CFD simulation of the hot channel of the IAEA, 10MW research reactors under FLOFA with the hot channel subject to a series of blockage scenarios was investigated by Salama et al [23]. Three adjacent channels were simulated and the flow is allowed to distribute among the three channels according to their hydraulic resistances. The phenomena of flow reversal were highlighted and investigated. Several important points were highlighted; probably the most interesting one is the complex interaction between the different flow and heat transfer mechanisms in the course of FLOFA in this system. Flow reversal phenomenon shows interesting behavior and was in direct connection with the different flow and heat transfer mechanisms. Furthermore, the peak clad temperature of the blocked channel implies that boiling was inevitably for this system particularly for higher blockage scenario (probably more than 20%). This indicates that even if this system was safe up to blockage ratio of, approximately, 85% under normal operating conditions, it wouldn’t been safe at all when FLOFA occurred and therefore more insight needed to be taken.

A reactivity insertion accident was simulated by ETRR2-RIA program developed by Khater et al [24] with a good degree of accuracy at different reactivity insertion scenarios. This program gave a wide flexibility for the reactor operator to simulate different behaviors of accident scenarios or any anticipated operational occurrence during reactor core management. The
compressed air, which caused a forced scram in ETRR-2 reactor is very important in accident with scram available scenario, because it can be overcome the initiating event, hence; no boiling will be predicted and the full shutdown margin is satisfied, while a little sub-cooled boiling was predicted in case of the stuck of control rod, or continue withdrawal. In transient without scram, a clad melt down was predicted in the hot channel. Generally, the ETRR-2 core withstood the uncontrolled withdrawal of a control rod when its first shutdown system was available and the lower conductivity of the oxide fuel $U_3O_8$, was resulted in a high temperature difference between the fuel and the clad, so this difference protected the clad from fast melting.

Research reactors with downward cooling and power above 10MW normally experience nucleate boiling in the event of a flow inversion after loss of power. Therefore, reactors of power more than 20MW were cooled by forced convection in the upward flow direction. However, a flow inversion phenomenon was predicted in a typical MTR reactor of 22MW with upward cooling by El-Morshedy [25]. The phenomenon was predicted by decreasing the decay heat multiplier and/or the pool temperature below the design value 40°C. MTRTHA code was updated and used for best-estimate simulation of the flow inversion phenomenon in the reactor core for two cases namely: two-flap valves open and one flap-valve fails to open. It was found that, at pool temperature equal 40°C, the coolant flow was inverted to the downward direction for a decay heat multiplier value of 0.6 for about 16 s before it inverted again to the upward direction for the two cases under study. The flow inversion phenomenon was also predicted for 8 and 35 s at pool temperatures of 35 and 30 °C, respectively in case of two flap-valves open leading to boiling for about 8 s in the average channel of pool temperature equal both 35 and 30 °C. While boiling continues for about 18 and 46 s in the hot channel for pool temperatures of 35°C and 30°C respectively; In case of one flap-valve fails, the flow inversion phenomenon was
predicted in the average channel for 13 and 39 sec. at pool temperatures of 35°C and 30°C, respectively. In this case boiling was predicted in the hot channel for the three pool temperature values for 6, 26 and 54 sec. for pool temperatures of 40°C, 35°C and 30°C, respectively. It was concluded that, the reduction of the decay heat value leaded to the prediction of flow inversion phenomenon without boiling in the reactor coolant channel where the heat generated in the core is low, while the reduction of the pool temperature value leads to the prediction of flow inversion phenomenon with boiling in the reactor coolant channels for long periods of time where boiling is undesirable phenomenon in this type of reactors as it could affect the aluminum clad integrity. Therefore, the installation of another flap-valve in each of the returning pipe to the core at a level of 4m above the core could be a solution to avoid the flow inversion phenomenon.

A model was presented by C. Housiadas [26] to predict with simple means the behavior of a small research reactor core during transient conditions. The model was based on a lumped parameters description of both the kinetics point model and thermal-hydraulic. The model was applied to the analysis of unprotected transients induced by an insertion of reactivity. The model provided a reasonably accurate means for determining the transient behavior of a research reactor, provided that no significant vapor generation had taken place in the core. The model was demonstrated to be applicable for various reactivity insertion rates, up to insertions as high as 1.5 $. The model could provide reasonably accurate prediction as long as the exit coolant temperature remains below saturation (no bulk boiling occurs in the core).

A dynamic model for the analysis of rapid core transients of U-Al dispersion fueled research reactors had been developed by Sofu [27]. The model included point neutron kinetics, two-phase thermal-hydraulics, and dimensional
heat conduction for average fuel temperatures. The flow and heat transfer regimes considered were single phase liquid flow, sub-cooled boiling, bulk boiling, film boiling, and single-phase vapor flow. Appropriate correlations were described for friction factor, heat transfer coefficient, void fraction, incipient boiling, and critical heat flux. The model developed was benchmarked against the RELAP5/Mod2 and REACC codes for verification of the thermal-hydraulics and neutronics/feedback behavior respectively. A series of comparisons with SPERT-II experiments were made to validate the coupled neutronics and thermal-hydraulic performance. The model was also applied to the core; a series of reactivity transients are analyzed related to control system malfunctions, pump starts in cold loop, and voids in regions with positive reactivity coefficient. The results indicated that the worst credible protected reactivity accident for the HFIR was due to optimum amount of void formation in the target region with positive reactivity coefficient. For this case, a partial hot plate melting was predicted in Mode-I at full power and at the end of fuel cycle conditions. However, additional scoping calculations suggested that fuel damage was limited to the hot plate only and the mechanical integrity of the rest of fuel elements was maintained. These calculations demonstrated that the model developed could be an effective tool in safety analysis and design of compact plate type fueled research reactors.

Wm. J. Garland et al [28] investigated heat transfer limitations of MNR, hand calculations; available correlations and CATHENA simulations consistently indicate that:
1. ONB is not a phenomena of concern since the sheath temperatures at ONB of about 125 EC are well below temperatures at which sheath blistering or swelling occur (400 to 450 EC).
2. Flow instability occurs at the onset of bulk boiling and can be reliably estimated given a good channel velocity estimate.
3. DNB follows somewhat after flow instability (perhaps at twice the power); hence bulk boiling can be conservatively used as a limiting condition for safety analysis.

A dynamic model of HANARO reactor was presented by Gee Y Han [29]. The reliable dynamic model of the reactor and its cooling systems was developed to perform the thermal-hydraulic analysis for transients. The developed dynamic model was properly implemented in the transient simulation code called H-SI, which used the DESIRE simulation language. The HSIM was intended for simulating efficiently the operational characteristics and the thermal-hydraulic behavior of HANARO for many transients. Through the thermal-hydraulic analysis to the power change, it was showed that the transient trends of the system safety parameters for power change were found to be consistent, the code was suitable for evaluating the thermal-hydraulic performance of HANARO, and could easily coupled with other component dynamic models of reactor such as the secondary coolant system model for integrated performance analysis.

Bokhari and Mahmood [30] analyzed the loss of flow accident (LOFA) of Pakistan research reactor-1 (PARR-1), which is one of the probable scenarios among other possible events such as reactivity-induced-accidents, loss of coolant accident, etc. The (PARR-1) reactor was initially converted from high enriched uranium (HEU) to low enriched uranium LEU fuel. The accident was assumed when the reactor is running at a steady-state power level of 9.8MW. Computer code PARET and standard correlations were employed to compute various parameters. Results predicted nucleate boiling in the core but the temperatures would remain far below the fuel clad melting point. They concluded that the proposed mixed fuel core for PARR-1 is safe against loss of flow accident. Although nucleate boiling would occur but the fuel clad
temperature only will rise up to 129°C, which is far below clad melting point of 580°C.

The heavy water moderated pool type reactor has a single, compact fuel assembly. After being unloaded from the reactor pool, such a fuel element has a high residual power of about 160 kW and thus has to be cooled sufficiently in the fuel storage pool. For this purpose, a passive cooling concept had been developed at by Baggemann et al [31]. The fuel element was placed at the storage pool in an exchange flask. The residual heat is removed from the exchange flask by internal natural circulation to the pool. The compact fuel assembly is about one meter high and consists of 280 evolving shaped plates which are arranged between two concentrically aluminum cylinders. The exchange flask consists primarily of an aluminum cylinder and a tube bundle heat exchanger. The heat exchanger is mounted on one side of the exchange flask with two supply lines, one above and one below the fuel element. The fuel element hangs inside the aluminum cylinder between these supply lines so that a natural circulation through the element and heat exchanger is established in normal operation, the residual heat is transferred to the fuel pool by the heat exchanger and cold water is fed to the lower plenum of the flask again. The aim of a numerical study performed to investigate the safety margins of this storage and cooling concept. For a subsequent accident analysis, it was important to gain an insight into each cooling channel of the fuel assembly and determine local effects, thus the fuel assembly has to be geometrically resolved. The model development was performed in 3 steps. In first sub-channel studies, the effects of grid resolution, wall treatment, turbulence and buoyancy model had been investigated on a single channel. As a second step, a porous medium had been implemented in order to get a first insight into the flow in the exchange flask and address the influence of the boundary conditions, the complete exchange flask is considered but the plates and the water channels were
modeled by a porous medium. Based on the results of these scoping tests, a final half symmetry model was built, resolving the sub-channels of the fuel element and the flow in the exchange flask.

For safety analyses to support conversion of MNSR reactors from HEU fuel to LEU fuel, a RELAP5-3D model was set up to simulate the entire MNSR system by Dunn et al [32]. This model included the core, the beryllium reflectors, the water in the tank and the water in the surrounding pool. Comparison with experimental data taken during commissioning of the NIRR-1 MNSR reactor in Nigeria shows that the steady-state and transient behavior of the HEU core is accurately predicted by the RELAP5-3D calculations. The same validated models and methodology were used for similar transients for LEU cores with two UO$_2$ fuel pin designs. Because of much higher melting temperatures of the LEU UO$_2$ fuel pellets and the zircaloy-4 cladding, the LEU cores will have significantly larger safety margins than the current aluminum-based fuel of the HEU core. Moderate uncertainties in the size of the as fabricated gap between LEU fuel pellets and the zircaloy-4 cladding are not important. Oxide fuel pellets should not crack at the low MNSR power levels. If cracking did occur, it would not be important. The steady-state thermal-hydraulic safety margins in the HEU and LEU cores are very large. Onset of nucleate boiling occurs at a power level of about 90 kW, approximately three times the maximum licensed power level. Onset of significant voiding was calculated to occur at about 350 kW – more than ten times the maximum licensed power level. Departure from Nucleate Boiling (DNB) would occur at a still higher power level. An LEU conversion demonstration of an MNSR reactor and a transient run with a reactivity insertion of ~ 4 mk is needed to validate the transient results.
A sub-channel analysis steady state thermal-hydraulic code (SACATRI) was developed for the Moroccan TRIGA MARK II research reactor by Merrouna et al [33]. The main objective of the thermal-hydraulic study of the whole reactor core was to evaluate the main safety parameters of the reactor core, and to ensure that they were within the safety limits for any operating conditions. The thermal-hydraulic model used in SACATRI was based on four partial differential equations that described the conservation of mass, energy and momentum. In order to assess the thermal-hydraulic mathematical model of SACATRI, this study focused on the quantification of the physical model accuracy to judge if the code was capable to represent the thermal hydraulic behavior of the reactor core with sufficient accuracy. The methodology adopted was based on the comparison between responses from SACATRI computational model and experimentally measured responses performed on the IPR-R1 TRIGA research reactor. The results showed good agreement between SACATRI predictions and the experimental measurements where the discrepancies observed (simulation experiment) are less than 6%.

A mathematical model was presented by Yamoah et al [34] to predict the reactor thermal-hydraulic behavior of Ghana research reactor-1. The model was based on a lumped parameter description and was completely defined by system of differential equations. The model was applied to study the effect of the cooling coils of the pool upper section on the reactor thermal-hydraulic parameters by modifying the model described in Zhang model; based on the results obtained; the maximum values of the reactor thermal-hydraulic parameters decreased when the cooling coil power was increased. The reactor began to lose the steady thermo-hydraulic state when the cooling coil power was just 30 kW. To have a neutronsically stable reactor, the cooling coil power should not be high as higher cooling coil power had consequences on the stability of the neutronics of the reactor. The model provided the understanding
of the influence of the cooling coil power, which cools the upper section of the pool, on various temperatures in the reactor.

The transient thermal-hydraulic behaviors of the JRR-3 under the normal operation condition had been investigated by Hirano et al [35] with using the THYDE-P1 code, focusing on the core flow reversal resulting from the transition from the forced circulation to the natural circulation. The thermal-hydraulic phenomena during the transient of interest are well understood by the THYDE-P1 calculation, which had clarified the important parameters affecting the local minimum DNBR and the peak fuel temperature. The following had been derived as the major results:

1- A sudden increase in the fuel temperature and a steep decrease in DNBR are calculated to occur at the core flow reversal.

2- The peak fuel temperature and the minimum DNBR depend strongly on the level of the decay power, namely, the time when the flow reversal occurs after the reactor shutdown. In the case of LOCA, these depend on the break area and the location. In the other cases, they depend only on; when the auxiliary pump is finally stopped.

3- The natural circulation valve was useful to ensure the flow path for the natural circulation. In order to limit the maximum valve diameter, it was useful to consider a type of sequence of events, such as a mis-opening of the valve under the possible condition with low flow.

The main objective of the reactor safety was to keep the reactor core in a condition, which wasn’t permitted any release of radioactivity into the environment. In order to ensure this, the reactor must have sufficient safety margins during all possible operational conditions (normal as well as accidental). To accomplish this, a study had been carried out by Bokhari and Mahmood [36] for the analysis of loss of flow accident (LOFA), which was one
of the probable scenarios among other possible events such as reactivity-induced-accidents, loss of coolant accident, etc. The study had been carried out for Pakistan research reactor, PARR-1, which was initially converted from HEU to LEU fuel. It is a swimming pool type reactor using MTR type fuel. Presently, a new core was proposed to be assembled containing LEU and some of the used (less burnt) HEU fuel elements. The accident was assumed when the reactor was running at a steady-state power level of 9.8MW. Computer code PARET and standard correlations were employed to compute various parameters. Results predicted nucleate boiling in the core but the temperatures would remain far below the fuel clad melting point.

Bsebsu and Bede [37] studied core thermal hydraulics design and analysis of Budapest nuclear research reactor (WWR-M type), which was a tank type, light water, cooled reactor with 36% enriched uranium coaxial annuli fuel. The Budapest nuclear research reactor was currently upgraded to 10MW of thermal power, while the cooling capacity of the reactor was designed and constructed for 20MW thermal power. This reserve in the cooling capacity served redundancy today but could be used for future upgrading too. The core thermal hydraulic design was therefore done for the normal operation conditions so that fuel elements might have enough safety margins both against the onset of nucleate boiling (ONB) not to allow the nucleate boiling anywhere in the reactor core and against the departure from nucleate boiling (DNB). Thermal hydraulic performance was studied, and it was shown that the 36% enriched UA1x-Al fuels in WWR-SM fuel coolant channel the possibility of force up the reactor power to 20MW_{th} was investigated with keeping the same core configuration and with three types of fuel coolant channel. The study was carried out for an equilibrium core, with compact load (223 fuel assemblies) under normal operation conditions.
The MIT Research Reactor (MITR) was in the process of conducting a design study to convert from High Enrichment Uranium (HEU) fuel to Low Enrichment Uranium (LEU) fuel. The currently selected LEU fuel design contains 18 plates per element, compared to the existing HEU design of 15 plates per element. A transitional conversion strategy, which consists of replacing three HEU elements with fresh LEU fuel elements in each fuel cycle, is proposed. Wang Y. et al. [38] analyzed the thermo-hydraulic safety margins and determined the operating power limits of the MITR for each mixed core configuration. The analysis was performed using PLTEMP/ANL, a program developed for thermo-hydraulic calculations of research reactors. Two correlations were used to model the friction pressure drop and enhanced heat transfer of the finned fuel plates: the Carnavos correlation for friction factor and heat transfer, and the Wong Correlation for friction factor with a constant heat transfer enhancement factor of 1.9. With these correlations, the minimum onset of nucleate boiling (ONB) margins of the hottest fuel plates were evaluated in nine different core configurations, the HEU core, the LEU core and seven mixed cores that consist of both HEU and LEU elements. The maximum radial power peaking factors were assumed at 2.0 for HEU and 1.76 for LEU in all the analyzed core configurations. The calculated results indicate that the HEU fuel elements yielded lower ONB margins than LEU fuel elements in all mixed core configurations. In addition to full coolant channels, side channels next to the support plates that form side coolant channels were analyzed and found to be more limiting due to higher flow resistance. The maximum operating powers during the HEU to LEU transition were determined by maintaining the minimum ONB margin corresponding to the homogeneous HEU core at 6MW. The recommended steady-state power was 5.8MW for all transitional cores if the maximum radial peaking is adjacent to a full coolant channel and 4.9MW if the maximum radial peaking was adjacent to a side coolant channel.
The conceptual thermal hydraulic design analyses for a 20MW reference AHR core had been jointly performed by Chae et al. [39]. The preliminary core thermal hydraulic characteristics and the safety margins for the AHR core were studied for various core flow rates and fuel assembly powers. Statistical method and the MATRA-h sub-channel code had been applied to evaluate the thermal hydraulic performances of the AHR core under the forced convection cooling mode during a nominal power operation and the natural circulation mode during a reactor shutdown condition. In addition, the safety margin evaluations were carried out for 2 typical accidents, a loss of flow accident by a primary pump seizure and a reactivity induced accident by a CAR rod withdrawal during a normal full power operation. Major design parameters for these analyses were minimum critical heat flux ratio (MCHFR), an onset of a nucleate boiling (ONB) margin and a maximum fuel temperature. Results of the thermal hydraulic analyses showed that the normal full power operation of the AHR could be ensured with a sufficient thermal margin for the onset of a nucleate boiling for a coolant velocity larger than 7.3 m/s. The AHR was also thought to have a sufficient natural circulation cooling capability up to at least 1.8MW to cool the core without the onset of a nucleate boiling in a channel after a normal reactor shutdown and during anticipated transients. It was also confirmed that the AHR core was sufficiently protected from the loss of a flow by a primary cooling pump seizure and the overpower transients by a CAR withdrawal from the MCHFR and fuel temperature points of view.

A simple model had been developed by Gaheen et al. [40] for the simulation and the analysis of MTR research reactors. To validate the developed model in transient conditions, the model was used to analyze the series of the IAEA 10MW benchmark transients. The model results for the analysis of reactivity insertion and loss of flow transients were compared with the calculations conducted in various institutions using different codes. The
proposed model provided an accurate prediction of the transient behavior where the clad temperature was less than the onset of nucleate boiling temperature. However, the power prediction was accurate except for those transients associated with extreme conditions in which boiling occurred in the average channel. This range of applicability covered most of the transient analysis requirements encountered in practice. It was concluded that the use of simple models was shown to provide useful capabilities for the analysis of most research reactors encountered in practice under transient conditions.

A computer code for the thermal–hydraulic analysis of plate type fuel reactors has been developed by Qing Lu. et al. [41]. To validate this code, the IAEA 10MW MTR benchmark transients were simulated. The calculated results of the Reactivity insertion accident (RIA) and loss of coolant accident (LOFA) transients were compared with those obtained in various institutions using different codes. Good agreement was achieved, which indicated that the models of this developed program were proper for thermal–hydraulic analysis of plate type research reactors. Furthermore, the partial and total blockages of one cooling channel were investigated with THAC-PRR code. The blockage occurred in one channel which was located in the middle of a single fuel assembly in the reactor core. Because of the channel blockage, the mass flow rate in this channel decreased and the temperatures increased, while the mass flow rate in the other channels increased. Coolant temperatures in the two adjacent channels also increased with that in the obstructed channel, while that in the rest of the channels decreased slightly. Thus, the fuel plates, which formed the obstructed channel were asymmetrically cooled. The results also demonstrated that no boiling occurred even when the channel was totally obstructed, except when the total blockage occurred in the hot channel, because the heat was transferred to its adjacent channels by lateral heat conduction in the fuel plate. This analysis underscored the importance of considering the impact
of the adjacent channels on the obstructed one, especially for plate type fuel reactors.

A dynamic model for the analysis of flow inversion and establishment of natural circulation core transient of LEU and HEU fueled research reactors had been developed by Kazeminejad [42]. The model included point neutron kinetics, single-phase thermal-hydraulics, and a one-dimensional heat conduction based on lumped parameters method. Appropriate correlations were described for heat transfer coefficient, friction factor and onset of nucleate boiling. The model developed is benchmarked against results obtained by other well established computational tools. Good agreement was obtained for the calculated peak clad temperature and all other key parameters. Recognizing the inherent differences between the models used in other computer codes, the overall agreement had been considered satisfactory in comparison with PARET results. The calculated maximum cladding surface temperature for the hottest reactor channel in both (HEU and LEU) cores did not exceed the allowable safety limit even during pump failure (and reactor shutdown) situations. It should be noted that some research reactors that are cooled by forced convection utilize downward coolant flow. In the event of pump failure and reactor shutdown, the reactor may be cooled by natural convection, which involves a reversal of the flow direction. The physical barrier that guards against uncontrolled radioactive releases is the cladding of the fuel elements, whose temperature is maintained below a certain limit (for example, onset of nucleate boiling) by cooling so that cladding integrity was ensured. The results demonstrated that the model developed in the study could be an effective tool in developing safety limits for the plate type fueled research reactor cooled by natural convection mode, which involved flow reversal. Also, the model allowed the simulation of all leading processes to the natural circulation. A major limitation of this model observed in the analysis of loss of flow transients
was the single-phase fluid flow model. Therefore, the model can be used for long period transients where boiling could be avoided.

A thermal hydraulic and safety analysis code-TSACC had been developed using Fortran 90 language by W. X. Tian et al. [43] to evaluate the transient thermal hydraulic behavior of the China advanced research reactor (CARR) under station blackout accident (SBA). For the development of TSACC, a series of corresponding mathematical and physical models were applied. Point reactor neutron kinetics model was adopted for solving the reactor power. All possible flow and heat transfer conditions under station blackout accident were considered and the optional correlations were supplied. The usual finite difference method was abandoned and the integral technique was adopted to evaluate the temperature field of the plate type fuel elements. A new simple and convenient equation was proposed for the resolution of the transient behaviors of the main pump instead of the complicated four-quadrant model. Gear method and Adams method were adopted alternately for a better solution to the stiff differential equations describing the dynamic behavior of the CARR. The computational result of TSACC showed the adequacy of the safety margin of CARR under SBA. For the purpose of Verification and Validation, the simulated results of TSACC were compared with those of RELAP5/MOD3 and a good agreement was obtained. The adoption of modular programming techniques enables TASCC to be applied to other reactors by easily modifying the corresponding function modules.

Maximo, et al. [44] developed a design of the emergency core cooling system (ECCS) for the IEA-Rlm pool type research reactor at Brazil. That system with passive features, used sprays installed above the core. The experimental program performed to define system parameters and to demonstrate the licensing authorities, that the fuel elements limiting temperature
was not exceeded was also presented. Flow distribution experiments using a core mock-up in fall-scale were performed to define the spray header geometry and spray nozzles specifications as well as the system total flow rate. Another set of the experiments using electrically heated plates simulating heat fluxes corresponding to the decay heat curve after full power operation at 5MW was conducted to measure the temperature distribution at the most critical position. The observed water flow pattern through the plates had a very peculiar behavior resulting in a temperature distribution which was modeled by a 2D energy equation numerical solution. They concluded that the design of the emergency core cooling system of the IEA-Rlm research reactor, upgraded to operate at 5MW used conservative parameters and redundancy criteria to assure that, in the occurrence of the postulated total loss of pool water, the maximum fuel temperatures was kept at safe levels. The spray flow distribution experiments defined the spray header specifications and demonstrated that every core component including the control elements were adequately irrigated by the spray system. The heated plate’s experiments demonstrated that during the first 30 min critical period, even under very conditions, the maximum temperatures were well below 500°C limit. In all tested conditions the measured temperatures were shown to be below the limiting value.

Khattab and Mena [45] analyzed core thermal-hydraulic for rod and plate types fuel elements without altering the core bundles square grid spacer (68 mm, side) and coolant mass flow rate. Coolant mass flux increased from 2000 Kg/m²s. Reactor power could be upgraded from 2MW to 10MW without significantly altering the steady state, thermal-hydraulic safety margins. Fuel, clad and coolant transient temperatures were determined inside the core hot channel during flow coast down using PARET code. Residual heat removal system of 20% coolant capacity was necessary for upgrading reactor power to encounter the case of pumps off at 10MW nominal operations.
A method was presented by Housiadas [46] to enables the analysis of loss of flow sequences, with scram disabled, in pool-type research reactors. The analysis was based on simulations performed with a customized version of the well-known coupled neutronics, thermal-hydraulics code PARET. The employed modeling approach was described in detail and the used assumptions thoroughly discussed. The obtained results could be regarded as typical of pool-type research reactors. Although GRR-1 reactor was used as a basis, core was exercised in the selection of parameters and ranges of variation, so to cover the usual research reactor cases. The simulations provided with a deep understanding of the mechanisms that determine the course of a loss of flow transient. The general physical picture obtained was the following, as flow rate was fell and reactor remained unprotected and coolant and fuel temperatures rise. The increasing temperatures induced a negative reactivity feedback and a corresponding power decreased. Eventually, reactor power was lowered enough, so that natural convection was sufficient to cool the core. An equilibrium situation was re-established in the reactor and the transient terminated without causing core damage. However, the safe transition from the initial to the final equilibrium state was interrupted if an unstable regime was encountered. This was connected with the development of massive bulk boiling in the hot channel, inducing flow instabilities of the Ledinegg type. The stability boundaries had been determined in terms of initial reactor power, initial pool temperature, peaking factor, and flow-decay time constant.

Experiments to understand the behavior of the nuclear reactors operational parameters allow improve model predictions, contributing to their safety. Developments and innovations used for research reactors can be later applied to larger power reactors. Their relatively low cost allows research reactors to provide an excellent testing ground for the reactors of tomorrow. The experiments described by Mesquita, et al [47] confirmed the efficiency of
natural convection in removing the heat produced in the reactor core by nuclear fission. The data taken during the experiments provided an excellent picture of the thermal performance of the IPR-R1 reactor core. The IPR-R1 TRIGA core design accommodated sufficient natural convective flow to maintain continuous flow of water throughout the core, which thereby avoided significant bubbles formation and restricted possible steam bubbles to the vicinity of the fuel element surface. The spacing between adjoining fuel elements was selected not only from neutronics considerations but also from thermo hydrodynamic considerations. The experimental data also provided information, which allowed the computation of other parameters, such as the fuel cladding heat transfer coefficient. The theoretical temperatures and mass flux were determined under ideal conditions. There was a considerable coolant cross flow throughout the channels. Note that the natural convection flow was turbulent in all channels near the centre. The temperature measurements above the IPR-R1 core showed that water mixing occurs within the first few centimeters above the top of the core, resulting in an almost uniform water temperature. The temperature at the primary loop suction point at the pool bottom depends on reactor power, as well as on the external temperature because it affected the heat dissipation rate in the cooling tower. The results could be considered as typical of pool-type research reactor. Further research could be done in the area of boiling heat transfer by using a simulated fuel element heated by electrical current (mock-up). The mock fuel element would eliminate the radiation hazard and allowed further thermocouple instrumentation. By using a thermocouple near the fuel element surface, the surface temperature could be measured as a function of the heat flux. It is suggested to repeat the experiments reported here, by placing a hollow cylinder over the core, with the same diameter of it, to verify the improvement of the mass flow rate by the chimney effect. These experiments could help the designers of the Brazilian research Multipurpose Reactor (RBM), which would
be a pool reactor equipped with a chimney to improve the heat removal from the core.

2.3 Heat exchangers and cooling towers review

A new heat mass transfer model was developed by Fahmy et al. [48] to predict the fouling process of calcium carbonate on heat transfer surface based on Kern-Seaton model. The effect of operating parameters like, time, concentration and fluid velocity on the scale formation of calcium sulphate had been taken into consideration in the present model. Also concentration effect on calcium sulphate deposition has been studied. Comparative results of the model with experimental data showed that the present model had the same trend and in good agreements with results investigated by others, so the new model could be credible to predict the fouling process. Also from the obtained results it was concluded that the scale formation in water is time dependant and had an asymptotic approach. The velocity of flow could reduce effectively the scale deposit due to the strong effect on the removal mechanism of the scale layer, and with increasing of water velocity in the secondary side the asymptotic value of sand deposition decreased.

Shell and Tube-type heat exchanger have wide application in nuclear industry where they play an important role in the transfer of heat from core to the heat sink; their cost minimization is an important target for both designers and users. A computer program for economical design of shell and tube heat exchanger using specified pressure drop is established by El-Fawal et al. [49] to minimize the cost of the equipment including the sum of discounted annual energy expenditures related to pumping. The design procedure depended on using the acceptable pressure drops in order to minimize the thermal surface area for a certain service, involving discrete decision variables. Also the proposed method took into account several geometric and operational
constraints typically recommended by design codes, and may provide global optimum solutions as opposed to local optimum solutions that were typically obtained with many other optimization methods. While fulfilling heat transfer requirements, it has anticipated to estimate the minimum heat transfer area and resultant minimum cost for a heat exchanger for given pressure drops. The capability of the proposed model was verified through two design examples. The obtained results illustrate the capacity of the proposed approach through using of a given pressure drops to direct the optimization towards more effective designs, considering important limitations usually ignored in the literatures.

The heat exchanger is the vital part of the cooling system of the TRIGA research reactor which was studied by Islam et al. [50]. The difference between the designed and operational thermal efficiencies had been found about 22%. The differences were due to changes the water properties with respect to operational hour of the reactor and the factors that are responsible to interrupt the secondary water quality. It was necessary to minimize the differences between the two. The increasing tendency of the pressure drop is due to several reasons was identified. Early mitigation of the increasing tendency of the pressure drop was needed to maintain the normal operation of the reactor. There were no maintenance activities performed after installation of the plate type heat exchanger for about six years. It was necessary to clean the heat exchanger in order to keep controlling pressure drop and temperature difference with in the design value limit. Proper maintenance of the heat exchanger could substantially improve the thermal performance over it and would protect any untoward situation in the primary and secondary cooling systems. It had been observed that the heat exchanger performances degraded a lot for the last 6 years although two preventive maintenance method i.e. two cycles of concentration and chemical injection method to the secondary water continued
time to time to restore the normal operating performance of the heat exchanger. It had been experienced that this two preventive action cannot restore the heat exchanger normal performance. That’s why it was recommended to disassemble the heat exchanger and then dipped the heat exchanger plates with sulfamic acids for several hours and finally clean the plates with brush and high jet of water. It was mentioned that heat exchanger was glue type which is not convenient to disassemble and mechanical wash. For that reason, recommendation was to install clip type heat exchanger which was very much easier for proper maintenance.

The heat exchangers had been studied experimentally by Sarker [51] to identify the optimum design parameters at the level of the best performance of a Hybrid Closed Circuit Cooling Tower (HCCCT) having a rated capacity of 1RT. In the standard designed condition, at cooling water having mass flow rate of 780 kg/h and at a WBT of 27°C, the thermal efficiency was found to be 0.3358 for the air flow rate of 2.75 m³/s. At an inlet air velocity of 2.75 m/s and wet bulb temperature (WBT) of 27°C, static pressure drop is measured to be 2.2 mmH₂O. Experimental study revealed that, for most cases in wet mode operation, a wet-bulb temperature of 27°C with a cooling water mass flow rate of 780 kg/h having an air velocity of about 2.75 m/s produced reasonable results especially with respect to the typical East-Asian meteorological constrains. The cooling capacity in wet mode operation was about 0.8RT (Refrigerant ton) which was within 23% of the rated capacity. In dry mode operation in the cooling capacity was about 0.3RT, it is evident that this poor capacity was mainly due to the absence of spray water and the HCCCT can be utilized in dry mode when the cooling requirement is lower and the economic operation is preferred. Results obtained from this study were supposed to provide experimental data which could be referred for the optimum design of the heat exchanger in a HCCCT.
A thermal acceptance test had been performed by El-Morshedy [52] at the ETRR-2 replacement cooling tower in order to evaluate its capability. The capability was not only the thermal acceptance criterion, but also used with the manufacturer performance curves to predict the cold water temperature at the off design points. Both the characteristic curve and performance curve methods were used to evaluate the results. The results indicated a capability of 106.5 % and 105.1 % for the two methods respectively. Since the capability of the cooling tower was higher than 90 %, the tower is thermally accepted. Based on the tests performed, the tower was capable of cooling the design water circulation rate times the capability value from 37°C to 30 °C at design wet bulb temperature of 24°C and design total fan driver output power of 48.1 hp.

A cooling tower basically consisted of three zones; namely, spray zone, packing and rain zones. In cooling towers, a significant portion of the total heat rejected may occur in the spray and rain zones. These zones were modeled and solved by Qureshi et al. [53] simultaneously using engineering equation solver (EES) software. The developed models of these zones were validated against experimental data. For the case study under consideration, the error in calculation of the tower volume is 6.5% when the spray and rain zones were neglected. This error was reduced to 3.15% and 2.65% as the spray and rain zones were incorporated in the model, respectively. Furthermore, fouling in cooling tower fills as well as its modeling strategy was explained and incorporated in the cooling tower model to study performance evaluation problems. The fouling model was presented in terms of normalized fill performance index as a function of weight gain due to fouling. It was demonstrated that the model is asymptotic, which was similar to typical asymptotic fouling model used in conventional heat exchangers.
Exergetic analysis is conducted on cooling towers and evaporative heat exchangers by Qureshi et al. [54]. In this regard, two definitions for the second-law efficiency were used. All computations were conducted with an Engineering Equation Solver (EES) program that had built-in functions for thermodynamic and transport properties. These built-in properties made it possible for a complete equation to be used for all streams and not only reduced the computational effort, but avoided the need for approximate solutions. For different input variables investigated, it was shown that the efficiencies were often increased or decreased monotonically. It was also noticed that an increase in the inlet wet bulb temperature consistently increased the second-law efficiency of all the systems investigated. It was shown that Bejan’s definition of second law efficiency was not limited in evaluating performance. The high second law efficiency of the evaporative heat exchangers indicated that, from thermodynamic standpoint, the processes occurring in these devices were approaching reversible. Furthermore, it was seen that the selection of the dead state properties and changes in it due to operation at off-design conditions did not significantly affected the overall efficiency of the system but it was noticed that the individual stream exergies were appreciably affected nonetheless.

The Egyptian Testing and Research Reactor no. 2 (ETRR-2) was commissioned at 1997 with maximum power 22MW for research purposes. An induced draft wet cooling tower counter flow type was put in operation since 2003 instead of the old one, and the investigations are achieved by Al Khatib et al. [55]. Merkel and Poppe analysis was applied to simulate the cooling tower packing to predict the water outlet temperature from the cooling tower and also to show the effect of ambient conditions on this temperature. Merkel number which is used to evaluate the cooling tower is also predicted, a Runge-Kutta numerical method was applied to solve the group of differential equations by using engineering equation solver program (EES). The results illustrated that
the cooling tower achieves good performance in various sever ambient conditions and maximum operating condition of reactor, the results showed also that at severe summer condition of wet bulb temperature equaled 24°C and cooling tower inlet temperature equaled 37°C the water outlet temperature equaled 30.2°C while the merkel number was predicted and equaled 1.253.

Kloppers and Kroger [56] investigated four natural draft cooling towers, which were not met their original design capability. The objective of the study was proposed to measure and reduce the water outlet temperature by at least 2 °C in order to meet or exceed the original design capability of these towers. The investigation to improve tower performance included thermal performance tests, outlet air mapping, visual inspection and a theoretical analysis. In order to meet the objective it is investigated whether small modifications and/or maintenance activities could be employed or whether the towers would have to be completely repacked with a newly designed water distribution system. Two possible alternatives were identified on how to achieve an additional two degrees centigrade cooling for four virtually identical cooling towers that were currently not met their design capability. The alternatives were identified after cooling tower performance test, outlet air mapping and a theoretical analysis. It was also investigated what the potential increase in tower performance would be if the air above the packing was increased by an external energy source. The performance of towers can be approximated by the WCTPE software if detailed performance charts were not available for the tower.

An experimental apparatus had been built by Caruso et al. [57] in order to perform sensitivity analyses on the performance of a natural draft-dry cooling tower. This component played an important role in the passive system for the residual heat decay removal foreseen in the MARS reactor and in the GCFR of the Generation IV reactors. The sensitivity analyses had investigated:
1) The heat exchanger arrangement; two different arrangements had been considered: a horizontal arrangement, in which a system of electrical heaters were placed at the inlet cross section of the tower, and a vertical arrangement, with the heaters distributed vertically around the circumference of the tower.

2) The shape of the cooling tower; by varying the angle of the shell inclination it was possible to obtain a different shape for the tower itself. An upper and a lower angle inclination were modified and by a calculation procedure eleven different configuration were selected.

3) The effect of cross wind on the tower performance. An equation-based procedure to design the dry-cooling tower was presented. In order to evaluate the influence of the shape and the heat exchanger arrangement on the performance of the cooling tower, a geometrical factor and a thermal factor were introduced. By analyzing the experimental results, engineering design relations were obtained to model the cooling tower performance. The comparison between the experimental heat transfer coefficient and the heat transfer coefficient obtained by the mathematical procedure showed that there was a good agreement. The influence of wind had been investigated on selected configurations and it had been studied by introducing several coefficients for the comparison: a wind effect and a pressure coefficients were used to evaluate the effect at the throat; a power coefficient, a mean static and mean dynamic pressure coefficients were also defined in order to compare the influence of the wind on different configurations. The obtained results showed that it was possible to evaluate the shape and the heat exchanger arrangement to optimize the performance of the cooling tower either in windless condition either in presence of cross wind.

A comprehensive mathematical model was developed by Papaef et al. [58] and used to examine the effect of various parameters on the thermal performance characteristics of a counter-flow wet cooling tower. The predictive
ability of the model was tested against pertinent experimental data obtained from the literature. An extensive parametric study was conducted aiming to scrutinize the influence of ambient air conditions, falling water mass flow rate and temperature on the variation of moist air thermodynamic properties, falling water temperature and evaporation rate inside the cooling tower. The effect of aforementioned parameters on the cooling capacity of the tower was assessed. The analysis of the theoretical results derived the following conclusions:

1. The increase of inlet air wet bulb temperature results in the increase of the overall change of dry bulb temperature of moist air. The affinity of air for water vapor absorption is amplified with reducing inlet wet bulb temperature. This favors the cooling capacity of the cooling tower and subsequently, its thermal efficiency because in this case the overall water temperature fall is increased due to the increase of the percentage of evaporated water mass. Hence, the profound interrelation between the degree of saturation of inlet air and the thermal performance of the cooling tower is acknowledged.

2. The increase of water to air mass ratio results in the enhancement of dry air temperature and humidity ratio rise due to increase of water vapor absorption rate from the air stream. However, reduction of the overall water temperature is observed with subsequent detrimental effects on the cooling capacity and thermal efficiency of the cooling tower because the air mass flow rate is invariable.

3. The increase of falling water temperature results in the increase of the overall dry bulb temperature and humidity ratio rise inside the cooling tower. It broadens also the total water temperature fall and the percentage loss of water mass due to evaporation. As a result, the thermal efficiency of the cooling tower is slightly increased.

The stepwise integration method is an approximation method for determining the water temperature in the fill layers for a counter-flow cooling
tower and in the fill cells for a cross-flow cooling tower was applied by Pirunkaset [59]. The value of \( \text{KaV} \) depended on the dynamics of the airflow and the distribution of water droplets above the top of the fill in the cooling tower. A given cooling tower can maintain a constant air flow rate and water flow rate. Accordingly, the magnitude of tower coefficient (\( \text{KaV} \)) of the fill also remains constant. The value of \( \text{KaV} \) can characterize the cooling tower and can be the basis for predicting the cooling tower performance under a given inlet wet bulb temperature. The effect of the inlet wet bulb temperature on the outlet water temperature and outlet dry bulb temperature at various values of \( \text{KaV} \) could be predicted by computer. When the inlet and outlet temperatures of the water, the flow rates of the entering water and air, and the inlet wet bulb temperature were known, it was possible to calculate the value of \( \text{KaV} \) by the stepwise integration method given the inlet and outlet temperatures of water, the flow rates of the entering air and water, the inlet dry bulb and inlet wet bulb temperatures and the hot water temperature were known, it was possible to predict the outlet water temperature by the stepwise integration method in accordance with condition 2 (given the value of \( \text{KaV} \) for the entire fill).

Water and air along the counter flow wet cooling tower is based on heat and mass transfer principles. The exergy analysis was used to explain the performance of simulated cooling tower. A method was presented by Muangnoi et al [60] for the prediction cooling tower performance by employing an exergy analysis. The method was validated using experimental data. The results show that Water exergy defined as the available energy carried by water to be supplied decreases continuously from top to bottom. For the air side, its exergy means the available energy of air to recover or utilize that supplied by water. There are two kinds of exergy in air that are due to exergy of air via convective heat transfer and exergy of air via evaporative heat transfer. It reveals that exergy of air is mainly controlled by exergy of air via evaporative heat transfer.
Exergy destruction was high at the bottom and reducing at the top. The distributions of exergy destruction could be used as a guideline to find optimal potential for improving cooling tower performance. For example, the use of a combination of two types of filling material was chosen by placing very efficient filling material with a large contact area at the bottom region where exergy destruction was high, and placing a regular one at the top region where exergy destruction was low. One important observation from this study was that the choice of the ambient conditions (eg. Wet bulb temperature and dry bulb temperature) affected the results of exergy analysis quite strongly. Currently, work was in progress to see the inlet conditions of air and water effects to the cooling tower performance.

This survey is used in deriving the equations of the proposed model and also to comprehend abnormal conditions and different accidents that face the MTR reactor.
CHAPTER (3)

MATHEMATICAL MODEL

In this chapter; the mathematical model is formulated to simulate the integrated cooling system of the reactor. The conservation equation of energy is the basic equation that is used to formulate our model, four main components constitute the model, and these components are reactor core, heat exchanger cooling tower and pipe lines. The model is firstly predicts the steady state condition of the reactor. The transient state employs a group of differential equations to simulate the thermal hydraulic behavior of the reactor; these equations are numerically solved by finite difference approximation with initial value problem consideration.

3.1 Distribution of heat generation in the fuel core

Based on the average thermal power of the reactor, it is possible to work out the heat generated per unit of volume at any point in the core. For a core which is geometrical cube, cosine distribution of the neutron flux (φ) is admissible in the direction of the three main axes. Considering that the fissile nuclei are uniformly distributed within the core and that the volumetric heat generation is proportional to the neutron flux, their variation in the core of the reactor may be expressed as follows El Wakil [61]:

\[ q(x, y, z) = q_m \cos \frac{\pi x}{2x_e} \cdot \cos \frac{\pi y}{2y_e} \cdot \cos \frac{\pi z}{2z_e} \]  \hspace{1cm} (3.1)

Where,

\( q_m \) : Maximum volumetric heat generation,

\( x_e, y_e, z_e \) are the extrapolated distances (distances from the center of the core to the point where neutron flux is nil).
In order to determine maximum heat generation, it is first necessary to calculate the average volumetric heat generation in the core \((q)\). In this case we know that the average heat flux density is provided by cengel [62]:

\[
\phi = \frac{W_{th}}{A_f \cdot N_{fe} \cdot N_{fp}}
\]

And we can calculate average volumetric heat generation in the core by:

\[
q = \frac{\phi}{x_f}
\]

Where,

- \(Q\) : Volumetric heat generation;
- \(W_{th}\) : Average thermal reactor power;
- \(A_f\) : Heating surface of each fuel plate;
- \(N_{fe}\) : Number of fuel element in the reactor;
- \(N_{fp}\) : Number of fuel plate per fuel element;
- \(\phi\) is the local heat flux based on cosine shape;
- \(x_f\) is the half of fuel plate thickness.

Then the value of maximum heat generation will be:

\[
q_m = q \cdot PPF
\]

Where,

- \(PPF\) : Power peaking factor.

Once the distribution of the volumetric heat generation is known, we can calculate the temperature of the fuel plate in the core at any position.

Temperature values are determined employing the hottest channel, which is located a distance of \(x=0, y=0\) from axis \(z\), so simplifying eqn. (3.1) we have

\[
q(z) = q_m \cdot \cos \frac{\pi z}{2z_e} \tag{3.2}
\]

The semi-active height of fuel core is \(z_c\), so the extrapolated height is calculated based on this relation:

\[
z_e = z_c + e
\]
Where $e$ is the extrapolated length

### 3.2 Reactor core model

The core model includes the determination of coolant, clad and fuel temperatures in both steady and transient states. The coolant channel is divided into a specified axial regions while the fuel plate is divided into a specified radial nodes, then a nodal thermal-hydraulic calculation for both average and hot channels is performed with a chopped cosine heat generation flux.

### 3.3 Steady state analysis

#### 3.3.1 Fuel analysis

Figure (3.1) shows a scheme of MTR fuel element with rectangular geometry. Heat flows in one dimension in $x$ direction. The flux $\phi$ and volumetric thermal source strength ($q$) are constant over the element cross-section, heat is conducted equally in $+x$ and $-x$ directions, and the mid plane $x = 0$ is the point of highest temperature.

![Fig. (3.1) View of the unit cell representing the MTR fuel plate.](image)

Considering only one half of the plane for heat flow in the $+x$ direction and a thin layer of thickness $(2x_f)$ at distance $(x_f)$ from the mid-plane, a heat balance for the case of steady state heat transfer can be derived.
Masoud [63] suppose that the thickness of the slab representing the fuel material is $2x_f$ and the thickness of slab representing each cladding is $x_{cl}$. The goal is to find the steady state ($\partial T/\partial t = 0$) temperature distribution in the slab assuming constant thermal conductivity ($\partial k/\partial T = 0$) for both fuel and cladding but with internal heat generation ($q \neq 0$) produced uniformly only in the inner slab (fuel).

3.3.2 Fuel temperature calculation

If the slab with internal heat generation ($q$) is made of fissile materials (referred to as fuel), sheath or cladding is used to contain the by-products of nuclear fission. The goal is to find the steady state temperature distribution in the slab assuming constant thermal conductivity for both fuel and cladding but with internal heat generation produced uniformly only in the inner slab (fuel). Due to symmetry, we only consider half of the composite slab in steady state with internal heat generation, the Fourier equation is applied, Masoud [53].

$$\nabla^2 T_f + \frac{q(z)}{k_f} = 0$$

$$\frac{d^2 T_f}{dx^2} + \frac{d^2 T_f}{dy^2} + \frac{d^2 T_f}{dz^2} + \frac{Q(z)}{k_f} = 0$$

The fuel temperature distribution is calculated by solving the general heat conduction equation for one dimension at steady-state condition:

$$\frac{d^2 T_f}{dx^2} + \frac{q(z)}{k_f} = 0$$

Rearrange the above equation to get,

$$\frac{d^2 T_f}{dx^2} = -\frac{q(z)}{k_f}$$

Applying the boundary condition, we have

The temperature gradient at fuel mid-point is vanish, so
\[ \frac{dT}{dx} = 0 \text{ at } x = 0 \text{ and} \]

the interfacial temperature between the fuel and clad is the same, so

\[ T_f(z, x_f) = T_{cl}(z). \]

The fuel temperature is then derived as:

\[ T_f(z, x) = \frac{q(z)}{2k_f}(x_f^2 - x^2) + T_{cl,i}(z) \]  \hspace{1cm} (3.3)

### 3.3.3 Clad temperature calculation

#### 3.3.3.1 Outer surface temperature

The Clad-surface temperature is calculated by applying Newton's cooling law Masoud [63]:

\[ \phi(z) = h_{conv} \cdot (T_{cl}(z) - T_c(z)) \]

Where \( h_{conv} \) is the local convective heat transfer coefficient

\[ \phi(z) = \frac{\phi_m \cdot \cos \frac{\pi z}{2z_c}}{h_{conv}} \]  \hspace{1cm} (3.4)

\[ \phi_m = \phi \cdot \text{PPF} \]

Where,

\( \phi \) is average heat flux,

\( \phi_m \) is the maximum admissible heat flux,

\( h_{conv} \) is the convective heat transfer coefficient.

The clad-surface temperature is predicted as:

\[ T_{cl,o}(z) = T_c(z) + \frac{\phi(z)}{h_{conv}} \]  \hspace{1cm} (3.5)

Where;

\( T_{cl}(z) \) : Outer clad-surface temperature,

\( T_c \) : Local temperature of coolant.

The maximum clad-surface temperature can be obtained by applying maximum heat flux (\( \phi_m \)).
3.3.3.2 Inner surface temperature

Bearing in mind the hypotheses already stated that:

1. The reactor operating under stationary conditions
2. Uni-directional heat flow

The following heat conduction equation by Fourier has been applied

\[ \varphi(x) = -k_{cl} \frac{dT}{dx} \]

Where,

\[ \frac{dT}{dx} \] is the temperature gradient in the thickness direction of fuel plate.

\[ dT = -\frac{\varphi(x)}{k_{cl}} dx \]

Integrating this equation, we have

\[ \int_{x_i}^{x_o} dT = -\frac{\varphi(x)}{k_{cl}} \int_{x_i}^{x_o} dx \]

After integration and rearrange we have,

\[ T_{cl,o} - T_{cl,i} = -\frac{\varphi(x)}{k_{cl}} x_{cl} \]  \hspace{1cm} (3.6)

This is the equation used to calculate the clad interior and exterior temperatures, and it the maximum temperature occurred along the z axis can be predicted.

3.3.4 Coolant temperature

It is assumed that the heat generated is transferred via convection from the fuel plate to the coolant, with no heat transmission via conduction existing in a lengthwise direction along the fuel plate. The amount of heat generated per unit time in an elemental volume of fuel core, situated at a distance \( z \) from the origin of the coordinates is:

\[ dQ_g(z) = q(z) \cdot A_{th} \cdot dz \]

Where;

\( A_{th} \) : Cross sectional area of the fuel; and
q(z) : Local volumetric heat generation given by:

\[ q(z) = q_m \cdot \cos \frac{\pi z}{2 \cdot z_e} \]

Where \( q_m \) is the maximum volumetric heat generation (at the core center).

The amount of heat removed by the coolant per unit time is:

\[ dQ_r(z) = w \cdot c_p \cdot dT \]

Where

\( w \) : Coolant mass flow rate and
\( dT \) : Temperature rise of the coolant.

At steady state condition, the amount of heat generated should be equal to the amount of heat removed,

\[ dQ_g(z) = dQ_r(z) \]

\[ dT = \frac{q_m \cdot A_{th}}{w \cdot c_p} \cdot \cos \frac{\pi \cdot z}{2 \cdot z_e} \cdot dz \]

\[ \int_{T_i}^{T_e} dT = \frac{q_m \cdot A_{th}}{w \cdot c_p} \int_{z_i}^{z_e} \cos \frac{\pi \cdot z}{2 \cdot z_e} \cdot dz \]

By integrating both sides, we obtain the distribution function for the average temperature of the coolant in a lengthwise direction along the channel.

\[ T_e(z) = T_i + \frac{q_m \cdot A_{th}}{w \cdot c_p} \cdot \frac{2 \cdot z_e}{\pi} \left[ \frac{\sin \frac{\pi \cdot z}{2 \cdot z_e}}{z_e} \right]_{z_i}^{z_e} \quad (3.7) \]

\( T_i \) is the coolant inlet temperature,
\( z \) is the direction of fuel element height.

**3.4 Calculation of the heat transfer coefficient**

In order to decide which flow is turbulent or laminar, a Reynolds number is calculated based on the velocity of coolant in the centerline of the coolant channel where \( z=0 \), so the velocity equation is expressed by:
\[ v(z=0) = \frac{w}{\rho A_{ch}} \]

\( w \) : Mass flow rate;
\( \rho \) : coolant density.

The calculation illustrates that the Reynolds number equals 46157, which is greater than 3000, the flow condition is defined as turbulent, and the heat transfer coefficient is calculated based on correlation of Dittus and Boelter equation [64].

\[ \text{Nusselt}(\text{Nu}) = 0.0243R_e^{0.8}Pr^{0.4} \quad (3.8) \]

The range of applicability includes \( 0.7 < \text{Pr} < 160, \text{Re} > 10,000, \) and \( \text{L}/D_e > 10. \) The Reynolds number is expressed as:

\[ R_e = \frac{D_h v}{\gamma} \quad (3.9) \]

Where

- \( R_e \) : Reynolds number,
- \( D_h \) : Hydraulic diameter of coolant channel,
- \( v \) : Average coolant velocity in centerline of coolant channel,
- \( \gamma \) : Kinematic viscosity of coolant and equals,

\[ \gamma = \frac{\mu}{\rho} \]

Where;

- \( \text{Pr} \) : Prandtl number,
- \( \mu \) : Dynamic viscosity of the coolant.

In our case the coolant channel have a rectangular configuration and we have to use the hydraulic diameter which is expressed by:

\[ D_h = \frac{4 A_{ch}}{p} \]

Where

- \( A_{ch} \) is the cross sectional area of the coolant channel,
p is the perimeter of the coolant channel.

\[ \text{Nu} = \frac{h_{\text{conv}} \cdot D_h}{k_c} \]  

(3.10)

3.5 Transient-state

In the transient heat transfer, the temperature normally varies with time as well as position. In the special case of variation with time but not with position, if the temperature of the medium changes uniformly with time, cengel [62] such heat transfer systems are called lumped systems. It is assumed the fuel clad region is lumped and the parameter under consideration like fuel, clad and coolant temperatures are lumped parameter, also this methodology is applied on the heat exchangers, pipes and cooling tower, the system of differential equations is solved numerically by explicit finite difference method, in the core model, fuel temperature of the fuel plate is predicted considering it as a mean value applying average heat flux, corresponding clad temperature is predicted assuming it the mean clad surface temperature.

The first step in establishing a criterion for the applicability of the lumped system analysis is to define characteristic length as

\[ L_c = \frac{V_f}{A_{ps}} \]

Where,

- \( L_c \): Characteristic length,
- \( V_f \): Fuel plate volume,
- \( A_{ps} \): Convective surface area.

And the Biot number (Bi) is defined as

\[ \text{Bi} = \frac{\text{conduction resistance within body}}{\text{convection resistance at the surface of the body}} = \frac{L_c/k}{1/h_{\text{conv}}} \]  

(3.11)
The Biot number is the ratio of the internal resistance of a body to heat conduction to its external resistance to heat convection. The smaller the Biot number, the accurate the lumped system analysis, it is generally accepted that lumped system analysis is applicable if 

\[ Bi \leq 0.1 \]

The calculation shows a Biot number less than 0.1 and equals 0.05 which means that the lumped analysis is satisfactory and the accuracy is accepted. The lumped-thermal capacity method is a convenient means of solving transient conduction problems.

The term lumped implies that temperature distribution in the object is not a concern since the entire object is represented by only one temperature. This method of analysis is useful when we need to estimate the temperature response of an object suddenly exposed to a different temperature. In reactor operations, the term “transient” describes, in general, a deviation from the normal value of one or more of the operating system temperatures and pressures, thermal power level, coolant flow due to any equipment failure. Under normal operating conditions, a balance is achieved between the production of heat and removal of heat by the coolant. However, if the energy production rate exceeds the removal capability, the excess energy is stored in the reactor core and coolant. It causes them to overheat and system may fail due to thermal loading. Possible failures due to energy mismatch depend upon the initiating event and accident sequence.

We may directly derive the heat conduction equation by using the differential analysis for heat diffusion in an infinitesimal control volume. The analysis is performed in the Cartesian coordinate system is:
\[
\left( \frac{\partial \Phi_x}{\partial x} \right) + \left( \frac{\partial \Phi_y}{\partial y} \right) + \left( \frac{\partial \Phi_z}{\partial z} \right) + q = \left( \frac{\partial \rho \cdot u}{\partial t} \right)
\]

Where,

\[
\left( \frac{\partial \Phi_x}{\partial x} \right) + \left( \frac{\partial \Phi_y}{\partial y} \right) + \left( \frac{\partial \Phi_z}{\partial z} \right)
\] is the rate of heat transfer by conduction mechanism, $q$ : Rate of internal heat generation per unit volume, and

\[
\rho \frac{\partial u}{\partial t}
\] : Rate of change of internal energy of control volume.

Our assumptions concern with one dimension coordinate, and a metallurgical bonding between the fuel core and aluminum cladding, as produced during the process of co-lamination to which the fuel plate is subjected is perfect with thermal resistance being nil, therefore, the temperature of both sides of core-cladding interface is the same.

Fig. (3.2) Fuel plate control volume.

Where,

\[ \Phi \] :Rate of heat flux,
In our model, the thermal properties can be considered constant over the temperature range of interest; consider a unit cell of MTR fuel element with rectangular geometry illustrated in Figure (3.2), a one-dimensional calculation is considered including a rectangular cooling channel of gap thickness 2d. The temperature for the local coolant temperature is $T_{co}$; clad temperature is $T_{cl}$ and fuel temperature is $T_f$.

The energy equation is applied on the core to predict the various region temperatures, incompressible turbulent flow of velocity $v$ is assumed to take place in the channel, transient heat balance for the local coolant, clad and fuel are investigated as follows

### 3.5.1 Fuel temperature

Studying the general conservation equation of energy for a control volume of fuel plate, the most frequently used form is, Massoud [63];

$$\sum_i w_{in} \cdot H_{in} + \sum q = \sum W_s + P \cdot V + \sum w_{out} \cdot H_{out} \quad \frac{d(M \cdot u)}{dt}$$

In our case, we consider that,

$$\sum_i w_{in} \cdot H_{in} = \sum w_{out} \cdot H_{out} = \sum W_s = P \cdot V = 0$$

Where in this equation, the summation of $q$ includes three major terms;

a- The rate of heat addition to control volume from external sources,

b- The rate of heat generation in the control volume from all internal sources;

and

c- The rates of heat removal from the control volume.

$$\sum q = \sum \left( \text{Rate of heat addition from external sources} \right) + \sum \left( \text{Rate of Internal heat} \right) - \sum \left( \text{Rate of heat removal from the control volume} \right)$$
The only summation $q$ includes the heat generated from Uranium fuel plate, in addition to heat removed by conduction heat transfer from the inner fuel meat towards the clad till the coolant which is expressed as:

Rate of heat addition from external sources = 0

Rate of internal heat generation = $q \cdot PPF$

$q$ : internal heat generation per unit volume,

Where, $q = \phi / x_f$

The rate of heat removal from control volume equals the conductive heat transfer from centerline of Uranium plate to the aluminum clad and equals ($qr_{f-cl}$)

$$qr_{f-cl} = \left( U / (x_f + x_{cl}) \right) \left( \hat{T}_f(z) - \hat{T}_{cl}(z) \right)$$

Where, $H$ : Enthalpy, $\hat{T}$ : Local temperature across z axis, $qr_{f-cl}$ : Heat transferred by conduction from fuel center to clad surface, $U$ : Overall conductive heat transfer coefficient, $x_f$ : Half thickness of fuel plate, $x_{cl}$ : Thickness of Aluminum clad ; and

Rate of change in internal energy in fuel region = \[
\frac{d(\rho_f \cdot c_{pf} \cdot \hat{T}_f)}{dt}
\]

Thus equating the above terms we derive the transient heat transfer equation of lumped fuel,

\[
\frac{d(\rho_f \cdot c_{pf} \cdot \hat{T}_f(z))}{dt} = \left( \frac{\phi}{x_f} \right) \text{PPF} - \frac{U}{x_f + x_{cl}} \left( \hat{T}_f(z) - \hat{T}_{cl}(z) \right)
\]  \hspace{1cm} (3.12)

The mean (core-averaged) value of an axially dependent quantity like coolant temperature (T) which follows the lumped analysis is given by
\[ T(t) = \frac{1}{z_{z_e}} \int_{z_e}^{z} \hat{T}(z, t) \, dz \]

So,

\[ \frac{dT}{dz} = \frac{1}{z_c} \left( T_{out} - T_i \right) = 2 \left( \frac{T_c - T_i}{z} \right) \]  

By applying the above rule on the eqns. (3.12), we obtain the mean values of the considered variables, the axial temperature distribution of the fuel in term of mean temperature is:

\[ \frac{d(\rho_f \cdot c_{pf} \cdot T_f(z))}{dt} = \left( \frac{\varphi}{x_f} \right)_{\text{PPF}} \cdot \frac{U}{x_f + x_{cl}} \left[ T_f(z) - T_{cl}(z) \right] \]

\[ \rho_f \cdot c_{pf} \cdot \frac{dT_f}{dt} = \left( \frac{\varphi}{x_f} \right)_{\text{PPF}} \cdot \frac{U}{x_f + x_{cl}} \left[ T_f(z) - T_{cl}(z) \right] \]  

Where,

\( \varphi \) : Average heat flux;

\( T(z) \) : Local temperature along the z axis;

\( \rho_f \) : Fuel density;

PPF: Power peaking factor;

\( c_{pf} \) : Fuel specific heat at constant pressure;

Subscripts f and cl denote the fuel and clad.

### 3.5.2 Aluminum clad analysis

In order to derive the lumped clad temperature, we assume the heat transferred from fuel to clad is transferred to coolant by convection, so the amount of heat transfer by convection \( (q_{\text{conv}}) \) is expressed by:

\[ q_{\text{conv}} = h_{\text{conv}} \cdot A_{cl} \left[ \hat{T}_{cl}(z) - \hat{T}(z) \right] \]
$h_{\text{conv}}$ is convective heat transfer coefficient that is calculated in steady state analysis. The rate of change in internal energy in clad surface temperature that is in contact with the coolant is expressed as:

\[
\frac{d(\rho_{cl}c_{pcl}\hat{T}_{cl})}{dt} = \left(\text{Conductive heat transfer from fuel to clad}\right) - \left(\text{Convective heat transfer to coolant channel}\right)
\]

So the transient heat transfer equation of lumped clad becomes

\[
\frac{d\left(\rho_{cl}c_{pcl}\hat{T}_{cl}\right)}{dt} = \frac{U}{x_f + x_{cl}} \left[\hat{T}_f(z) - \hat{T}_{cl}(z)\right] - \frac{h_{\text{conv}}}{b} \left[\hat{T}_{cl}(z) - \hat{T}_c(z)\right]
\]

(3.15)

Applying the rule in eqn. 3.15 we have,

\[
\frac{d(\rho_{cl}c_{pcl}T_{cl})}{dt} = \frac{U}{x_f + x_{cl}} \left[T_f(z) - T_{cl}(z)\right] - \frac{h_{\text{conv}}}{b} \left[T_{cl}(z) - T_c(z)\right]
\]

(3.16)

Where,

$\rho_{cl}$ : Clad density,
$c_{pcl}$ : Clad specific heat at constant pressure;

b : Fuel plate width;

U : Overall heat transfer coefficient and is given by

\[
\frac{1}{U} = \frac{x_f}{k_f} + \frac{x_{cl}}{k_{cl}}
\]

Where,

T : Mean temperature,

$x_f$ : Half fuel plate thickness,

$x_{cl}$ : Clad thickness,

$k_f$ : Fuel thermal conductivity, and

$k_{cl}$ : Clad thermal conductivity.
3.5.3 Coolant analysis

The transient analysis of coolant channel is based on heat transfer by convection that is transferred completely to the coolant channel that comes from the core inlet through channel with velocity \(v\) until leaves the channel, so

\[ q_{cc} = \rho_c \left( \frac{v}{z_c} \right) c_{pc} \left( \hat{T}_c (z) - T_i \right) \]

Where,
\( \rho_c \) : Coolant density,
\( q_{cc} \) : heat transferred to coolant channel by convection,

Then the transient heat transferred equation through coolant channel is expressed by

\[
\frac{d}{dt} \left( \rho_c c_{pc} \hat{T}_c \right) = \left( \frac{h_{conv}}{b} \right) \left[ T_{cl} (z) - \hat{T}_c (z) \right] - \rho_c v c_{pc} \frac{d\hat{T}_c (z)}{dz}
\]

Using the equation (3.13), we conclude the equation of coolant in form of mean values.

\[
\rho_c c_{pc} \frac{d(T_c)}{dt} = \left( \frac{h_{conv}}{b} \right) \left[ T_{cl} (z) - T_c (z) \right] - 2 \rho_c v c_{pc} \frac{\hat{T}_c (z) - T_i}{z}
\]

(3.17)

So,

\[ T_c = \frac{(T_{out} + T_i)}{2} \]

Where,
\( \rho_c \) :Coolant density,
\( c_{pc} \) :Coolant specific heat at constant pressure.
\( v \) :Coolant velocity inside the channel, and
\( b \) :Half channel width.

Where \( T_c \) is the mean coolant temperature, \( T_{out} \) the outlet coolant temperature, and;

\[ T_{out} = \hat{T}_c (z = z_c, t), \]
$T_{out}$ : Coolant outlet temperature,
$T_i$ the inlet coolant temperature, where;

$$T_i = \hat{T}_c(z = 0, t)$$

$T_f$, $T_{cl}$ are the mean fuel and clad temperatures. Assume that the shape of the above functions remains unchanged with time when we investigate maximum values and it is identical to the profile corresponding to static conditions, considering the steady-state solutions of Eqs. (3.14), (3.16) and (3.17), it can be shown that the axial profiles can be expressed as follows, Housiadas [26].

$$\hat{T}_{cl}(z) = \hat{T}_c(z) - (T_{cl} - T_c) \cos \left( \frac{\pi(z - (z_c/2))}{z_e} \right)$$

(3.18)

$$\hat{T}_f(z) = \hat{T}_{cl}(z) + (T_f - T_{cl}) \cos \left( \frac{\pi(z - (z_c/2))}{z_e} \right)$$

(3.19)

On the other hand, Eq. (3.18) can be used to determine the maximum fuel temperature point. By differentiating Eq. (3.19) with respect to $z$, equating it to zero, and solving for $z$, the following equation for the axial location of the maximum fuel temperature is obtained as:

$$z_{max} = \frac{z_c}{2} + \frac{z_e}{\pi} \tan^{-1} \left( \frac{2}{\pi} \frac{z_e}{z_c} \frac{(T_c - T_i)}{(T_f - T_c)} \right)$$

(3.20)

$z_e$ : Extrapolated height,

### 3.6 Heat exchanger model

A transient during heat exchanger operation is generally caused by temperature change in inlet or outlet of cold stream or hot stream; these
changes come from reactor power change or from loss of heat sink result from fan failure.

Inlet and outlet temperatures of coolant to inlet and exit from heat exchanger may also change by other factors but our model deals only with the changes come from loss of heat sink. In this analysis we consider counter flow heat exchangers and divide the exchanger along its length to several nodes. Both streams are assumed to be incompressible and local fluid properties are used. By explicitly modeling the plate region, thermal inertia of the plate material would then appear in the formulation. Shown in Figure (3.3) is the schematic of a counter flow heat exchanger, divided into n nodes but only three nodes are shown. Node n, for example, receives mass and energy from node n-1, as carried by the mass flow rate of the hot stream and, in turn, delivers mass and enthalpy to node n+1. Due to the liquid incompressibility, mass flow rate into node n equals the mass flow rate into node n+1, as only energy would accumulate in node n. There is also a transverse energy transfer out of node n of the hot stream, through the plate surface into node n of the cold stream.

The hot stream enters at an enthalpy of \((H_{h,in})\) and leaves at an enthalpy of \(H_{h,out} < H_{h,in}\). If the heat exchanger is fully insulated so that there is no heat loss to the environment, then under transient-state operation hence, the
energy balance in the axial direction for element n in the hot stream yields, Cengel [62]:

\[ q_{hs} - q_s = \frac{\partial}{\partial t} \left( M_h \cdot c_{pw} \cdot T_i \right) \]

\[ q_{hs} = w_{w,h} \cdot c_{pw} \cdot (T_{h,in} - T_{h,out}) \quad (3.21) \]

Where \( q_s \) is the quantity of heat transferred axially from hot stream to plate surface of heat exchanger, this heat is transferred to cold stream through the surface and is dependent on the heat transfer area that separates hot and cold stream.

The heat transferred across the surface is defined based on logarithmic mean temperature difference and expressed by the following relation

\[ q_s = U_{hx} \cdot A_{hx} \cdot \Delta T_{LMTD} \quad (3.22) \]

Where

\( U_{hx} \) is the overall heat transfer coefficient,
\( A_{hx} \) is the heat transfer area,
\( \Delta T_{LMTD} \) is the logarithmic mean temperature difference.

This heat transferred from hot stream to the heat exchanger surface area is transferred completely to the cold stream; this leads us to define the heat transferred by cold stream by the following equation

\[ q_{cs} = w_{w,c} \cdot c_{pw} \cdot (T_{c,out} - T_{c,in}) \quad (3.23) \]

So the transient relation of the cold stream follows the following equation

\[ q_s - q_{hs} = \frac{\partial}{\partial t} \left( M_c \cdot c_{pw} \cdot T_i \right) \]

The general form of equations in transient state is simplified to the following two main relations as follows
\[ M_{w,h}c_{pw}\frac{dT_{h,\text{out}}}{dt} = w_{w,h}c_{pw}(T_{h,\text{in}} - T_{h,\text{out}}) - U_{hx}A_{hx}\Delta T_{\text{LMTD}} \quad (3.24) \]

\[ M_{w,c}c_{pw}\frac{dT_{c,\text{out}}}{dt} = U_{hx}A_{hx}\Delta T_{\text{LMTD}} - w_{w,h}c_{pw}(T_{c,\text{out}} - T_{c,\text{in}}) \quad (3.25) \]

The finite difference formulation of the above equation is according to the following equations,

\[ M_{w,h}c_{pw}\frac{T_{h,\text{out}}^{i+1} - T_{h,\text{out}}^{i}}{\Delta t} = w_{w,h}c_{pw}(T_{h,\text{in}}^{i} - T_{h,\text{out}}^{i}) - U_{hx}A_{hx}\Delta T_{\text{LMTD}} \quad (3.26) \]

\[ M_{w,c}c_{pw}\frac{T_{c,\text{out}}^{i+1} - T_{c,\text{out}}^{i}}{\Delta t} = U_{hx}A_{hx}\Delta T_{\text{LMTD}} - w_{w,c}c_{pw}(T_{c,\text{out}}^{i} - T_{c,\text{in}}) \quad (3.27) \]

Where,

\[ \Delta T_{\text{LMTD}} = \frac{(T_{h,\text{in}} - T_{c,\text{out}}) - (T_{h,\text{out}} - T_{c,\text{in}})}{\ln((T_{h,\text{in}} - T_{c,\text{out}})/(T_{h,\text{out}} - T_{c,\text{in}}))} \]

### 3.7 Cooling tower model

The cooling tower consists of six independent units, each unit have its own fan and packing. The energy balance of control volume deals with the packing area only which has the major part of heat transfer, neglecting the other area of spray zone and rain zone. Air is drawn from the bottom of the tower by the fan fixed on the top of cooling tower and the water is forced to fall from the top of the tower to down on the fill packing, the heat is transferred from the hot water to the contact air direct contact, Kern [65].

Consider a control volume of cooling tower in the Figure (3.4), and applying energy balance between input and output heat we find,

Rate of change in internal energy=energy inlet – energy outlet

General form of energy balance gives the following equation

\[ \frac{d(M,H)}{dt} = \sum_{i} w_{i}H_{i} - \sum_{i} w_{out}H_{out} \]

After simplifying the equation becomes
Fig. (3.4) Control volume of cooling tower

\[
\frac{d(MH)_w}{dt} = \left( w_{w, in} . H_{w, in} + w_a . c_p . T_{a, in} \right) - \left( w_{w, out} . H_{w, out} + w_a . c_p . T_{a, out} \right)
\]

\[
M_w \frac{dH_w}{dt} = \left( w_{w, in} . H_{w, in} + w_a . c_p . T_{a, in} \right) - \left( w_{w, out} . H_{w, out} + w_a . c_p . T_{a, out} \right)
\]

So, change of water enthalpy is expressed by in term of temperature difference

\[ H = c_p dT \]

Considering that the inlet mass flow rate is the same as outlet mass flow rate, then

\[
M_w \frac{dT_w}{dt} = \left( w_w . H_{w, in} + w_a . c_p . T_{a, in} \right) - \left( w_w . H_{w, out} + w_a . c_p . T_{a, out} \right) \tag{3.28}
\]

The finite difference formulation of the above equation is written as

\[
M_w \frac{T_{w, out}^{n+1} - T_{w, out}^n}{\Delta t} = \left( w_w . H_{w, in}^n + w_a . c_p . H_{a, in}^n \right) - \left( w_w . H_{w, out}^n + w_a . c_p . H_{a, out}^n \right) \tag{3.29}
\]
The enthalpy of the air-water vapor mixture per unit mass of dry air is expressed by Kröger[66],

\[ H_{a,\text{in}} = c_{pa} \cdot T_{a,\text{in}} + \omega (H_{fg} + c_{pv} \cdot T_{a,\text{in}}) \] (3.30)

Where:
\( \omega \) : Humidity ratio,
\( H_{fg} \) : Latent heat of vaporization,
\( c_{pa} \) : Specific heat of dry air, and
\( c_{pv} \) : Specific heat of saturated water vapor.

And \( c_{pa} \) and \( c_{pv} \) are evaluated at \((T+273.15)/2 \) °k according as follows, Kröger[58]

\[ c_{pa} = 1.045356 \times 10^3 - 3.161783 \times 10^{-1} T + 7.083814 \times 10^{-4} T^2 - 2.705209 \times 10^{-7} \] (3.31)

And.
\[ c_{pv} = 1.3605 \times 10^3 + 2.31334T - 2.46784 \times 10^{-10} T^5 + 5.91332 \times 10^{-2} T \times 10^{-13} T^6 \] (3.32)

The latent heat of vaporization is given according to Kroger [66] by:

\[ H_{fg} = 3.4831814 \times 10^6 - 5.8627703 \times 10^3 T + 12.139568 T^3 \] (3.33)

\[ - 1.40290431 \]

Where,
\( T_w \) : Coolant temperature in cooling tower,
\( T_{a,\text{in}} \) : Air wet bulb temperature,
\( MW \) : Mass of coolant that the cooling tower contains,
\( w_a \) : Air mass flow rate,
\( c_{pa} \) : Air specific heat at constant temperature.

### 3.8 Pipe line model

Applying the energy equation on the control volume shown in Figure (3-5) of the pipe line we deduce the following
\[
\frac{\partial}{\partial t} \left( M_{p,cv} \cdot c_p \cdot T \right) = q_{in} - q_{out}
\]

\[
\frac{\partial}{\partial t} \left( M_{p,cv} \cdot c_p \cdot T \right) = w_p \cdot c_p \left( T_{p,out} - T_{p,in} \right)
\]

\[
\left( M_{p,cv} \cdot c_p \right) \frac{dT}{dt} = w_p \cdot c_p \left( T_{p,out} - T_{p,in} \right) \tag{3.34}
\]

The finite difference formulation of the above equation is

\[
M_{p,cv} \left( \frac{T_{p,out}^{n+1} - T_{p,out}^n}{\Delta t} \right) = w_p \left( T_{p,out}^n - T_{p,in}^n \right) \tag{3.35}
\]

Where,

\( M_{p,cv} \) is the mass of coolant that pipe line contains and equals \( M_{p,cv} = \rho \cdot A_p \cdot L \)

Where,

\( A_p \) is the cross sectional area of the pipe,

Where,

\[
A_p = \frac{\pi}{4} \cdot D_p^2
\]

\( L \) : Length of the pipe,

\( w_p \) : Mass flow rate in the pipe,

\( D_p \) : Pipe inner diameter.
3.9 Thermal-hydraulic safety margins

3.9.1 Onset of Nucleate Boiling

The convective heat transfer is strongly dependent on the hydraulics, notably on velocity and flow regime, as well as on the material properties. MTR operates exclusively in single phase liquid mode under normal operation. The coolant is normally highly sub-cooled, even near the fuel sheath surface. If coolant flow is impaired sufficiently or if power should rise sufficiently, the coolant-sheath interface temperature will rise to or above the saturation temperature of the coolant; 117ºC in this case Wm. J. Garland [67].

The onset of nucleate boiling (ONB) occurs before the onset of bulk boiling and that elevated sheath temperatures occur at ONB, further investigation is warranted to ensure that fuel damage does not occur at thermal hydraulic conditions below bulk boiling.

The condition under which boiling will be initiated in the coolant channel is when the clad temperature equals to or exceeds the onset of nucleate boiling temperature, \( T_{ONB} \), as follows

\[
T_{ONB} = T_{sat} + (\Delta T_{sat})_{ONB}
\]

Where \((\Delta T_{sat})_{ONB}\) is given by Bergles and Rohsenow correlation, Mesquita [59] which is valid for water only over the pressure range 1-138 bar. It predicts the onset of nucleation with water flowing at velocities up to 17.5 m/s.

\[
(\Delta T_{sat})_{ONB} = 0.556\left[\frac{\varphi_{ONB}}{1082.9^{1.156}}\right]^{0.463P_{0.0234}}
\]

Where \(P\) is the local pressure in bar and \(\varphi_{ONB}\) is in W/m².

The heat flux that causes nucleation at a wall superheat \((\Delta T_{sat})_{ONB}\) and a system pressure, \(P\) is

\[
\varphi_{ONB} = h_{conv} \cdot [T_{ONB} - T_{co,h}]
\]
3.9.2 Onset of flow instability

The term “flow instability” refers to flow oscillations of constant or variable amplitude. Flow oscillations are undesirable phenomenon for several reasons. First, sustained flow oscillation may cause undesirable forced mechanical vibration of components, second, flow oscillations may cause system control problems, which are of particular importance in water cooled reactors where the coolant also acts as moderator and third, flow oscillations affect the local heat transfer characteristics and the boiling crisis. The limiting heat flux at the onset of flow instability is calculated using the developed correlation Zacarias [68].

\[
\varphi_{OFI} (W/m^2) = \left[ R c_p \rho \cdot V \frac{b \cdot d}{b_h \cdot (z_c)} (T_{sat} - T_{in}) \right] 
\]

Where \( R \) follows the relation \( R = \frac{1}{D_h} \left( \frac{1}{1 + \eta \frac{L}{D_h}} \right) \)

and \( \eta \) is the bubble detachment parameter = 25,

\( T_{ONB} \): Onset of nucleate boiling temperature,

\( T_{sat} \): Fluid saturation temperature, \( \sim 117 \) °C

\( T_{in} \): Coolant inlet temperature,

\( b \): Fuel plate width,

\( b_h \): Heated width,

3.9.3 Departure from nucleate boiling

In addition to onset of nucleate boiling and flow instability, the investigated departure from nucleate boiling (DNB) is performed. The correlations of Labuntsov were found to be the best suited to low pressure
plate type research reactors (MTR). The Labuntsov correlation states Mesquita[68]:

\[
\varphi_{\text{DNB}} = 1.454.0(p)^{1/2}(1 + 2.5V^2/\theta(p))^{1/4}[1 + (15.1/P^{1/2})(C_{p.\Delta T_{sub}}/H_{fg})] 
\]

(3.40)

Where,

\[
\theta(p) = 0.99531.P^{1/3}(1 - \frac{P}{P_{cr}})^{4/3} 
\]

\[
\varphi_{\text{DNB}} : \text{Departure from nucleate boiling heat flux,} \\
P : \text{Pressure at the axial location, bar} \\
P_{cr} : \text{Critical pressure of the coolant, bar} \\
\Delta T_{sub} : \text{Coolant sub-cooling at the axial location of CHF (heated length exit).} \\
\]

This correlation was strictly applicable to velocities above 2 m/s.

3.9.4 Determination of hydraulic stability of the fuel plate.

In order to determine the coolant flow velocity at which the parallel plates that form a flow channel collapse, the formula developed by Miller [60] is applied.

\[
v_c = \left[ \frac{c.g.E_m.x_r.d}{\rho.b.(1 - v^2)} \right] 
\]

(3.41)

Where,

\[
v_c : \text{Critical velocity of the coolant;} \\
c : \text{Constant which depends on support condition for fuel plate;} \\
g : \text{Gravitational acceleration;} \\
E_m : \text{Elasticity modulus for aluminum material;} \\
v : \text{Poisson modulus for Aluminum material;} \\
d : \text{Half channel thickness, and} \\
b : \text{Fuel plate width.} \]
Parallel fuel plate vibrations have been analyzed by Miller who derived the above formula for critical velocity based on the interaction between the changes in cross-sectional areas, coolant velocities and pressures in two adjacent channels. When a high speed flow passes through a narrow passage, the pressure head of the stream is converted into a velocity head, which creates a suction force on the wall which makes the narrow gap walls to be decreased and stops the flow.

In order to determine the maximum coolant velocity, the mass flow formula is applied, it is evident that with constant flow, the maximum coolant flow speed will be reached when the transversal area of the channel and the density of the coolant are their minimum level. In order to know this last value, it is necessary to determine the temperature of this coolant at the upper end of the fuel core, employing the next equation.

\[ T_{\text{out}} = T_i + \frac{q_{m,A_{\text{th}}}}{w \cdot c_{pw}} \cdot 2 \cdot z_e \int_{-z_e}^{z_e} \frac{x}{\pi} \sin \left( \frac{\pi x}{2 \cdot z_e} \right) \]  \hspace{1cm} (3.42)

The water density is calculated based on the outlet coolant temperature in equation (3.42), therefore, the mass flow rate equation is used to get the maximum coolant velocity.

\[ v_{\text{max}} = \frac{w}{\rho \cdot A_{tc}} \]

Where,

A_{tc} : Transversal area of the channel.

**3.10 Two-dimensional steady state of fuel plate surface**

In this section a two dimensional sub-model is conducted in order to obtain a two dimensional temperature distribution through the fuel plate surface.
Consider a rectangular region of fuel plate in which heat convection is significant in the y and z directions. Now divide the y-z plane of the region into a rectangular mesh of nodal points spaced $\Delta y$ in the direction of the width of the fuel plate and $\Delta z$ in the direction of heated length, apart in the y and z directions, respectively; as shown in Figure (3.6), and considers a unit depth of x in the x-direction; Cengel [62]. Our goal is to determine the temperatures at the nodes, and it is convenient to number the nodes and describes their position by the numbers instead of actual coordinates. A logical numbering scheme for two-dimensional problems is the double subscript notation $(i, j)$ where $i=0, 1, 2, \ldots, n$ is the node count in the y-direction and $j=0, 1, 2, \ldots, m$ is the node count in the z-direction. The coordinates of the node $(i, j)$ are simply $y = i \times \Delta y$ and $z = j \times \Delta z$, and the temperature at the node $(i, j)$ is denoted by $T_{i,j}$.

Now consider a volume element of size $\Delta y \times \Delta z \times 1$ centered about a general interior node $(i, j)$ in a region in which heat is generated defined as $Q$, and the thermal conductivity $k$ is constant. Again assuming the direction of heat
conduction to be toward the node under consideration at all surfaces, the energy balance on the volume element can be expressed as

\[
\begin{align*}
\text{(Rate of heat conduction)} & \quad \text{(Rate of heat generation inside)} & \quad \text{(Rate of change of energy content of the element)} \\
\text{(At the left, top, right, and bottom surfaces)} & \quad \text{(the element)} & \quad \text{(of the element)} \\
(\varphi_{\text{cond, left }} A_{cr}) & + (\varphi_{\text{cond, top }} A_{ct}) & + (\varphi_{\text{cond, right }} A_{ce}) & + (\varphi_{\text{cond, bottom }} A_{cb}) \\
+ (q.V)_{\text{element}} & = \frac{\Delta E_{\text{element}}}{\Delta t} & = 0 \\
\end{align*}
\]

Since the energy content of a medium (or any part of it) does not change under steady conditions and thus \(\Delta E_{\text{element}} = 0\).

For the steady state case; assuming the temperatures between the adjacent nodes to vary linearly and noting that the heat transfer area is \(A_z = \Delta y \times 1 = \Delta y\) in the y-direction and \(A_y = \Delta z \times 1\) in the z-direction, the energy balance relation above becomes

\[
\begin{align*}
\frac{k.\Delta x.\Delta z. T_{i-1,j} - T_{i,j}}{\Delta y} & + \frac{k.\Delta x.\Delta z. T_{i,j+1} - T_{i,j}}{\Delta z} & + \frac{k.\Delta x.\Delta z. T_{i+1,j} - T_{i,j}}{\Delta y} \\
+ \frac{k.\Delta x.\Delta z. T_{i,j+1} - T_{i,j}}{\Delta z} & + q_{i,j} \Delta x.\Delta y.\Delta z & = 0 \\
\end{align*}
\]

Dividing each term by \(k.\Delta y.\Delta z\) and simplifying gives

\[
\begin{align*}
\frac{T_{i-1,j} - 2T_{i,j} + T_{i+1,j}}{(\Delta y)^2} & + \frac{T_{i,j+1} - 2T_{i,j} + T_{i,j+1}}{(\Delta z)^2} & + \frac{q_{i,j}}{k_{el}} = 0 \\
\end{align*}
\]

Where \(\Delta y\) is the direction of fuel width, and \(\Delta z\) in the direction of heated length

\(\Delta y = b / (n-1)\) and \(\Delta z = z / (m-1)\) where \(n\) is the number of nodes in width direction and \(m\) is a number of nodes in the heated length direction.

Applying this equation in the clad region, we have,
\[
\left[ \frac{T_{c(i-1,j)} - 2T_{c(i,j)} + T_{c(i+1,j)}}{(\Delta y_{c})^2} \right] + \left[ \frac{T_{c(i,j-1)} - 2T_{c(i,j)} + T_{c(i,j+1)}}{(\Delta z)^2} \right] = 0 \quad (3.43)
\]

The above two dimensional equations are used to predict the temperature contour at the fuel plate surface area attached to the coolant, the boundary condition of the solution is considered to be convection state at all direction, (left, right, bottom and top), in the left and right directions, it is the same condition according to the following equation:

\[
\varphi_{\text{node}} = h_{\text{conv}} \left( T_{c(i,j)} - T_{\infty} \right) \quad \text{(Temperature at left direction)} \quad (3.44)
\]

\[
\varphi_{\text{node}} = h_{\text{conv}} \left( T_{c(i,j)} - T_{\infty} \right) \quad \text{(Temperature at right direction)} \quad (3.45)
\]

Where \( q_{\text{node}} \) is the thermal heat flux on the node,

On the top and bottom surfaces the convective heat transfer equations are

\[
\varphi_{\text{node}} = h_{\text{conv}} \left( T_{c(i,j)} - T_{\text{out}} \right) \quad \text{(Temperature at the top direction)} \quad (3.46)
\]

\[
\varphi_{\text{node}} = h_{\text{conv}} \left( T_{c(i,j)} - T_{1} \right) \quad \text{(Temperature at the bottom direction)} \quad (3.47)
\]

Where \( T_{\text{out}} \) is temperature of the coolant at the core exit, and \( T_{1} \) is the temperature of the coolant at the core inlet.
CHAPTER (4)

Model Assessment

In this chapter the present model is validated against the PARET code for steady-state operation and verified by the reactor operation records for transients. The records acquisitions are collected from MTR at different regimes.

4.1 Steady-state

The steady-state results of the present model for the MTR are compared with the PARET code [70]. PARET is a coupled neutronics and thermal-hydraulics computer code developed at Argonne National Laboratory. It is designed for use in predicting the course and consequences of nondestructive reactivity accidents in small reactor cores. The comparison between the present model and PARET code is performed for the typical core configuration of 29 fuel elements under a nominal power of (22 MW) and a power peaking factor of 3. The steady-state core flow rate is taken as 1900 m$^3$/h. Figure (4.1) and Figure (4.2) illustrate the temperature profiles predicted by the present model and the PARET code for coolant, clad and the fuel temperatures along the average and hot channels respectively; Considering the design conditions, coolant enters the reactor at temperature of 40ºC, and increases along heated length according to the heat deposition values in both channels, and leaves the average and hot channel at 50ºC and 63ºC respectively. The clad surface temperature distributions along the heated length are also shown and the predicted results show that the maximum values predicted are 65.5ºC and 97.5ºC for both the average and hot channels respectively at distance equals 48 cm from the reactor core inlet. The fuel-center temperature distributions show maximum values of 75ºC and 117.0ºC for the average and hot channels respectively.
Fig. (4.1) Temperature profile along the average channel.

Fig. (4.2) Temperature profile along the hot channel.
Table [4.1] shows the maximum temperature values predicted by the present model and PARET code for both the average and hot channels. The results show a good agreement between the model and the PARET code results for steady-state operation.

<table>
<thead>
<tr>
<th>Maximum Temperatures</th>
<th>Average channel</th>
<th>Hot channel</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Present</td>
<td>PARET</td>
</tr>
<tr>
<td>Coolant</td>
<td>50.0</td>
<td>49.8</td>
</tr>
<tr>
<td>Clad-surface</td>
<td>65.5</td>
<td>66.5</td>
</tr>
<tr>
<td>Fuel-center</td>
<td>75.0</td>
<td>76.8</td>
</tr>
</tbody>
</table>

4.2 Transient-state

In transient-state, the model is verified against reactor operational records. The comparison includes core inlet temperature, core outlet temperature, cooling tower outlet temperature and cooling tower inlet temperature. Two available operational states are selected with (19.69MW) reactor thermal power with two different ambient conditions, one in winter season with recorded wet bulb temperature value of (14°C) and relative humidity value of (42%) and the other in the summer season with recorded and measured wet bulb temperature value of (22°C) and relative humidity value of (52%). Figure (4.3) illustrates both of the reactor core inlet and outlet temperatures compared with the operational records. The power is raised by automatic reactor protection system from (16.0MW) to (19.69MW) at wet bulb temperature (14°C) and the transient state is predicted until reaching the steady state, coolant enters the core at (24°C) during (16MW) representing the initial thermal power and increased by increasing the reactor
power to reach (28.6°C) at a power of (19.69MW). In a similar way, the core exit temperature increased from (32°C) to (36.2°C). In the other hand, the results are predicted again at the same thermal power and a wet bulb temperature of (22°C). It is found that the predicted core inlet temperature increases from (29°C) to (36.1°C) and the exit core temperature increases from (37.2°C) to (45.1°C).

Fig. (4.3) Core inlet and outlet temperatures
Fig. (4.4) Cooling tower inlet and outlet temperatures.

Fig. (4.5) Heat exchanger cold and hot streams outlet temperatures.
Figure (4.4) shows the cooling tower inlet and outlet temperatures under the same conditions. In case of wet bulb temperature of (14°C), it is shown that water enters the cooling tower at a temperature of (17.5°C) at a power of (16MW) and this value increases by increasing the power to reach (21°C) at a power of (19.69MW). In similar manner water exits the cooling tower at (23.4°C) and this value increases to reach (28.2°C). In case of a wet bulb temperature of (22°C), water enters the cooling tower at (23.8°C) and exits at (27.5°C) at a power of (16MW), while these values increased to (25.6°C) and (34°C) respectively at a power of (19.69MW).

Figure (4.5) illustrates the predicted and recorded temperatures of hot and cold stream of the heat exchanger during operation the reactor at wet bulb
temperature (22°C) and thermal power (19.69MW), hot stream fluid goes to the reactor core inlet and cold stream fluid outlet goes to the cooling tower inlet. The hot stream temperature value equals (36.5°C) while the cold stream temperature value equals (32.7°C), the predicted temperatures is also found in accepted values during ascending the state from transient to steady state.

![Core inlet and outlet temperatures](image)

**Fig. (4.7) Core inlet and outlet temperatures**

Figure (4.6) illustrates the different conditions of ambient temperature and its effect on the cooling tower outlet temperature, three ambient conditions are recorded and identified by wet bulb temperature, (14°C), (18°C), (22°C), and the thermal power is recorded 19.69MW. The results show that increasing the wet bulb temperature increases the cooling tower outlet temperature, a
good agreement is noticed between the prediction results and the recorded data.

Figure (4.7) illustrates the core inlet temperature at various operating thermal power and different wet bulb temperature, the results show a good agreement between the predicted results and the recorded data.
In this chapter the proposed model is applied on Egyptian Testing and Research reactor number 2 (ETRR-2) under abnormal situation due to loss of the ultimate heat sink. The Engineering Equation Solver (EES) [71] is used to solve the model differential equations.

Fig. (5.1) Flux distribution in the average and hot channels.
Calculation is performed for the loss of heat sink transient due to cooling tower failure at different operating conditions with varieties of safety actions implemented by reactor protection system.

5.1 Core heat flux profile at design condition

The flux distributions in both average and hot channels during steady-state normal operation at the reactor nominal power of 22MW are presented in Fig. (5.1); the heat flux profile is considered chopped cosine shape along longwise direction; maximum heat flux is achieved in the center of the fuel plate. The shape of the heat flux used in the present model is established by an analytical function (cosine profile) in terms of the ratio of the peak flux to the average flux in the core distribution. This peaking factor is referred to as the hot channel factor. Figure 5.1 shows also a scheme of the channel cell and the coordinate system used to define the analytical function. The maximum heat flux values are equal to 38E+4W/m² and 117E+4W/m² for both the average and hot channels respectively. During the loss of heat sink the heat flux profiles in both the average and hot channels are considered as shown in Fig (5.1) until Scram triggering. The heat flux value is equal to the steady-state values multiplied by the corresponding value of the normalized power vs time after shut-down of the ANS [72] curve (appendix (2)).

5.2 Reactor core steady state

The Figures (5.2), (5.3) show results of steady state thermal behavior of reactor core in regime I during stabilizing the reactor thermal power at 11MW.
Fig. (5.2) Temperature profile along the average channel

Fig. (5.3) Temperature profile along the hot channel.
Core fuel, clad and coolant temperatures are illustrated in both average and hot channels along the fuel plate active length; the table illustrated below depicts the maximum temperatures fulfilled in both cases.

Table [5.1]: Maximum temperature values predicted by the present model in regime I.

<table>
<thead>
<tr>
<th>Maximum Temperatures</th>
<th>Average channel</th>
<th>Hot channel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coolant</td>
<td>50.13°C</td>
<td>62.74°C</td>
</tr>
<tr>
<td>Clad-surface</td>
<td>56.75°C</td>
<td>77.21°C</td>
</tr>
<tr>
<td>Fuel-center</td>
<td>60.45°C</td>
<td>85.44°C</td>
</tr>
</tbody>
</table>

### 5.2.1 Core temperature profile at design condition

The present model is employed to predict the temperature distribution in the fuel plate along the radial direction at the plate center point, the generated heat is assumed to be at the design reactor thermal power.

Figure (5.4) illustrates the predicted temperature profile along the fuel-plate center for both the average and hot channels corresponding to applicable maximum heat flux, where the maximum fuel and clad surface temperatures are found to equal 118°C and 98.6°C respectively in hot channel. In the other hand, the predicted maximum temperatures of both the fuel and clad-surface in the average channel are found to equal 74°C and 66°C respectively.
5.3 Reactor start-up simulation

The reactor start-up transient is simulated for the two operation regimes (I) and (II). The maximum thermal power is restricted by (11MW) in regime (I) and (22MW) in regime (II). The design parameters are considered for core inlet temperature of 40°C and the wet bulb temperature of 24°C during the simulation.

5.3.1 Regime (I) simulation

In regime (I), the reactor start-up is simulated from (100kW) thermal power to (11MW) using only one branch of the core cooling system includes one plate type heat exchanger, one operated pump and one pump in reserve. Figure (5.5) and Figure (5.6) show the calculated maximum temperature values during start-up in regime (I) for both the hot and the average channels respectively.
Figure (5.5) Maximum hot channel fuel, clad and coolant temperatures during start-up for regime I.

Figure (5.6) Maximum average temperatures of fuel, clad and coolant during start-up for regime I.
The steady state condition is reached and the calculated maximum hot channel temperature values predicted in regime (I) for fuel-center, clad-surface and coolant are 85.53°C, 78.12°C, and 62.43°C respectively. For the average channel, the maximum temperature values for fuel-center, clad-surface and coolant are 59.7°C, 56.2°C and 49.2°C.

Figure (5.7) shows the core inlet and outlet temperature of coolant during start-up in regime I during start-up. The maximum values reached for regime (I) are 40°C and 49.2°C for the inlet and outlet respectively.
5.3.2 Regime (II) simulation

The power rising in regime (II) is assumed from (11MW) and it is assumed that the operator continues operating the reactor from steady state of regime I after operating the two cooling branches.

![Graph showing temperature values during start-up for regime II](image)

Fig. (5.8) Maximum hot channel fuel, clad and coolant temperatures during start-up for regime II

This means that, additional pump and heat exchanger are operated in parallel and so the core flow rate is doubled. Figure (5.8) and Figure (5.9) show the calculated maximum temperature values during start-up in regime (II) for both the hot and the average channels respectively. The maximum temperature values predicted in the hot channel for fuel-center, clad-surface and coolant are 117.2°C, 99.3°C and 64.1°C respectively.
Fig. (5.9) Maximum average channel fuel, clad and coolant temperatures during start-up for regime II.

While in average channel the maximum temperature values predicted for fuel-center, clad-surface and coolant are 74.4°C, 66°C and 49.9°C respectively. Figure (5.10) shows the core inlet and outlet temperature of coolant during regime (II). The maximum values reached for regime (II) are 40°C and 49.9°C for the inlet and outlet respectively. In the other hand, the cooling tower inlet and outlet temperatures of water in regime II are predicted and depicted in Fig. (5.11) The maximum values reached are 37.1°C and 30°C for inlet and outlet respectively. The temperatures predicted from cooling tower in regime I is the same as in regime (II) because the coolant flow rate in regime (I) is the half and the cooling tower in regime operates with its half capacity.
Fig. (5.10) Core coolant inlet and outlet temperatures during start up in regime II

Fig. (5.11) Cooling tower water inlet and outlet temperature during start up in regime II
5.4 Loss of heat sink simulation

The present model is used to simulate the loss of the reactor ultimate heat sink. Simulation is performed for two operational regimes; regime (I) and regime (II), in the regime (I) the reactor operated at a power (11MW), only one primary cooling branch with one pump and three cooling tower cells with one operated pump in the secondary cooling system are running. While in regime II, the reactor operates at a power (22MW), two primary cooling branches are operated with one pump in each branch and six cooling tower cells run with two operated pumps in the secondary cooling system operate.

The simulation of the heat sink failure is performed by excluding the cells of the cooling tower one by one respectively, the reactor is considered to operate at its design parameters for steady-state normal operation before the loss of the heat sink simulation occurred. The reactor protection system is designed to limit the reactor power to its eighty percent in case of reactor core inlet temperature reaches 43°C, only this emergency safety action is triggered in this case, if the safety system fails to reduce the reactor power to its 80% another safety action is recalled to shut down the reactor by emergency shutdown which is called scram, which is considered and classified as emergency shutdown state, it will shut down the reactor when the reactor core inlet temperature reaches 44°C.

5.4.1 Regime I simulation

The following events are considered in the simulation of the accident at regime I:
1-One cooling tower cell is failed.
2-Two cooling tower cells are failed.
3-Three cooling tower cells are failed.
Each event of the above mentioned is predicted to perform the time that the cooling system consumes to preserve and keep the reactor in operation before the safety action is recalled by reactor protection system.

Firstly, the reactor is assumed to operate normally in design conditions, so the steady state is dominant state before applying the loss of heat sink, this steady state lasts exactly 400 seconds during the simulation before transient analysis is performed.

Fig. (5.12) illustrates the prediction of cooling tower outlet temperature in regime I, these predictions are performed for less than three operated cooling tower cells during accidental events, one, two and three cell are excluded in order, the results show a rapid increase in tower outlet temperatures as fans excluded from the system, one failed fan leads to power reduction as the first emergency safety system at cooling tower outlet temperature equals 33.7°C and corresponding time equals 685 sec., while the emergency scram is taken place at other cases of two and three failed fans at temperature equals 35.2°C at times 570 sec., and 440 sec. respectively.

Figure (5.13) shows the variation of the core inlet temperature during loss of the heat sink due to one, two and three cells’ failure. The failure of one cell leads to a rapid increase of the core inlet temperature to reach the safety limit 43°C leading to a power reduction action by the reactor protection system (reduce the reactor power to its eighty percent of the present reactor power).

If the power reduction safety action is failed to be triggered the RPS call the scram emergency shutdown and this is shown in the second case while more fans are failed, Scram triggering due to high core inlet temperature (the reactor thermal power drops suddenly to shut down the reactor to its 7% of its reactor power). More time is reasonably consumed than power reduction time.
Figures (5.14), (5.15) and (5.16) show the variation of the maximum hot channel coolant, clad and fuel temperatures during loss of the heat sink due to one, two and three cells’ failure. After the loss of one cooling tower cell, the reactor inlet coolant temperature increased to reach a setting value of 43°C at time 840 seconds, then power reduction occurs and the coolant temperature decreases gradually and becomes in safe limit compared to the safety settings. Both the clad and fuel temperatures follow the same trend with higher values where a maximum values of 87.8°C, and 80.4°C in hot channel are predicted.

In a case of two and three cooling tower cells’ failure, the predicted maximum coolant temperature reaches the scram limit 54°C in core outlet temperature, and 44°C in the core inlet temperature, and the corresponding values of fuel and clad temperature are 88.9°C and 81.5°C, these events consume about 570 seconds for two cells’ failure and 450 seconds in case of three cell’ failure. The maximum temperature values of regime I are depicted in table [5.2].

Table [5.2]: Maximum temperature values predicted during loss of heat sink for regime I.

<table>
<thead>
<tr>
<th>Event.</th>
<th>Coolant</th>
<th>Clad</th>
<th>Fuel-center.</th>
<th>Time(sec)</th>
<th>Triggered</th>
</tr>
</thead>
<tbody>
<tr>
<td>One cell fails</td>
<td>64.7°C</td>
<td>80.4</td>
<td>87.8</td>
<td>680</td>
<td>P. R</td>
</tr>
<tr>
<td>Two cells fail</td>
<td>65.78°C</td>
<td>81.5</td>
<td>88.9</td>
<td>570</td>
<td>Emg. Scr.</td>
</tr>
<tr>
<td>Three cells fail</td>
<td>65.81°C</td>
<td>81.5</td>
<td>88.9</td>
<td>450</td>
<td>Emg. Scr.</td>
</tr>
</tbody>
</table>
Table [5.3]: Maximum temperature values of tower during loss of heat sink for regime I.

<table>
<thead>
<tr>
<th>Event</th>
<th>Tower outlet temperature</th>
<th>Tower inlet temperature</th>
</tr>
</thead>
<tbody>
<tr>
<td>One cell fails</td>
<td>33.7°C</td>
<td>39.3°C</td>
</tr>
<tr>
<td>Two cells fail</td>
<td>35.2°C</td>
<td>39.3°C</td>
</tr>
<tr>
<td>Three cells fail</td>
<td>35.2°C</td>
<td>39.3°C</td>
</tr>
</tbody>
</table>

Where,

P. R is the Power Reduction safety action, and

Emg. Scr is the Emergency Scram safety action.

Fig. (5.12) Cooling tower outlet temperature in regime I.
Fig. (5.13) Inlet coolant temperature variation during loss of heat sink for regime I

Fig. (5.14) Maximum coolant temperature during loss of heat sink for regime I
Fig. (5.15) Maximum clad-surface temperature variation during loss of heat sink for regime I.

Fig. (5.16) Maximum fuel-center temperature variation during loss of heat sink for regime I.
5.4.2 Regime II simulation

In Regime II simulation, the following events are considered respectively and the predicted temperatures are investigated one by one to estimate the time of temperature increase independently.

1- One cooling tower cell is failed.
2- Two cooling tower cells are failed.
3- Three cooling tower cells are failed.
4- Four cooling tower cells are failed.
5- Five cooling tower cells are failed.
6- Six cooling tower cells are failed.

Figure (5.17) illustrates the prediction of cooling tower outlet temperature in regime II, these predictions are performed for less than six operated cooling tower cells during accidental events, and one to six cells are excluded in order. The results show a rapid increase in tower outlet temperatures as fans excluded from the system, one failed fan still conserve the reactor operation close to safety setting but it still in safe condition, the outlet temperature of cooling tower equals 32.6°C, two failed fan leads to power reduction as the first emergency safety action at cooling tower outlet temperature equals 33.7°C, and corresponding time equals 530 sec., while the emergency scram is taken place at other cases of three till six failed fans.

Figure (5.18) shows the variation of the core inlet temperature of coolant during loss of the heat sink due to one to six failed cells. The failure of one cell leads to increase the core inlet temperature gradually to a value 42.9°C, but still in the safe record compared with the setting value, that is happened because of cooling tower good performance which capable to remove generated heat from reactor core, two failed cells are investigated and shown, the temperature increase more to reach the setting values 43°C in this case and the power reduction is triggered by RPS to reduce the power to eighty percent of the present power after 550 seconds, coolant, clad and fuel
temperatures are predicted and the values 43.02°C, 102.8°C, and (119.4°C) appears respectively. Scram is triggered due to high core inlet temperature 44°C in the later cases, 3, 4, 5, 6 failed cells.

Figures (5.19), (5.20) and (5.21) show the variation of the maximum coolant, clad and fuel temperatures in hot channel during loss of these cells due to one, to six cells failure in order. Investigation the case of three cooling tower cell failure’s is the beginning explanation of scram emergency shutdown, the reactor inlet coolant temperature increased to reach a setting value of 44°C at time 600 seconds, both inlet and outlet temperature in reactor core are designed to trigger the reactor protection system simultaneously to assure safety in the MTR reactors, then scram occurs while any of them is called. Both the clad and fuel temperatures follow the same trend with higher values than in the case of one and two cells’ failure, where a maximum values of 104°C, and 120.5°C in hot channel are predicted respectively. For a four, five and six cooling tower cells’ failure, the predicted maximum coolant admissible to reach the scram limit in shorter times, and the corresponding values of clad and fuel temperature values are depicted in table [5.4].

Table [5.4]: Maximum temperature values predicted during loss of heat sink for regime II.

<table>
<thead>
<tr>
<th>Event.</th>
<th>Coolant.</th>
<th>Clad</th>
<th>Fuel-</th>
<th>Time(sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>One failed cell</td>
<td>66.75°C</td>
<td>102.6°C</td>
<td>119.2°C</td>
<td>1500</td>
</tr>
<tr>
<td>Two failed cells</td>
<td>67.50°C</td>
<td>102.8°C</td>
<td>119.9°C</td>
<td>530</td>
</tr>
<tr>
<td>Three failed cells</td>
<td>68.6°C</td>
<td>104°C</td>
<td>120.8°C</td>
<td>600</td>
</tr>
<tr>
<td>Four failed cells</td>
<td>68.6°C</td>
<td>104°C</td>
<td>120.8°C</td>
<td>440</td>
</tr>
<tr>
<td>Five failed cells</td>
<td>68.6°C</td>
<td>104°C</td>
<td>120.8°C</td>
<td>310</td>
</tr>
<tr>
<td>Six failed cells</td>
<td>68.6°C</td>
<td>104°C</td>
<td>120.8°C</td>
<td>240</td>
</tr>
</tbody>
</table>
Fig. (5.17) Cooling tower outlet temperature in regime II.

Fig. (5.18) Core inlet temperature variation during loss of heat sink for regime II.
Fig. (5.19) Maximum coolant temperature variation during loss of heat sink for regime II

Fig. (5.20) Maximum clad-surface temperature variation during loss of heat sink for regime II
Fig. (5.21) Maximum fuel-center temperature variation during loss of heat sink for regime II.

5.5 Thermal-hydraulic safety margins

The essential goal of thermal-hydraulics is to preserve the fuel element integrity in order to avoid a radioactive release. Therefore, safety margins are adapted to achieve this goal. It can be defined as the ratio between the heat flux for the occurrence of an undesirable phenomenon and the maximum admissible heat flux corresponding to the hot channel. Table [5-5] shows the best-estimate minimum safety margins by the present developed model for the two operation regimes.
Table [5-5] Best-estimate minimum safety margins

<table>
<thead>
<tr>
<th>Margin</th>
<th>ONB</th>
<th>OFI</th>
<th>DNB</th>
<th>Vel.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Regime I: steady-state</td>
<td>2.686</td>
<td>5.79</td>
<td>6.784</td>
<td>7.92</td>
</tr>
<tr>
<td>Regime I: one to six cells</td>
<td>2.616</td>
<td>5.49</td>
<td>6.297</td>
<td>7.92</td>
</tr>
<tr>
<td>Regime II: steady-state</td>
<td>1.404</td>
<td>2.895</td>
<td>4.737</td>
<td>4.03</td>
</tr>
<tr>
<td>Regime II: one to six cells</td>
<td>1.319</td>
<td>2.745</td>
<td>4.397</td>
<td>4.03</td>
</tr>
</tbody>
</table>

5.6 Hydraulic stability of fuel plate

The critical velocity is calculated based on Miller correlation [60], for plates with long edges restrained by attachment to lateral plates, the constant k=15. The density of the coolant is considered corresponds to the density at 40°C, which is the inlet temperature in the fuel element, the calculated critical velocity is be found 19.8 m/sec.

The density of the coolant is calculated based on outer coolant temperature from fuel element is used to calculate maximum velocity, the upper end of the core(z=+40cm), this temperature equals 63°C, so for more outlet temperature less density is be found and leads to more velocities while considering constant flow rate. The results show that maximum velocities not exceed 5 m/sec and the velocity ratios are depicted in table [5.5], which means that it doesn’t present any risk to hydraulic stability.
5.7 Clad-surface temperature contour

Fig.(5.22) Temperature contour on clad surface of the average channel in regime I.

The two-dimensional clad-surface temperature contour is predicted and depicted in Figures (5.22) and (5.23) for regime II in both the average and
hot channels respectively. The plate width is divided into 20 nodes while the plate height is divided into 81 nodes, the node number (40) located at the core entrance while the node number (-40) located at the core exit. The maximum temperature values are predicted and equaled (98.4°C) and (66.4°C) for both the average and hot channels respectively. The maximum temperature values occur at distance 58.5 cm from the entrance.

![temperature contour on clad-surface of the hot channel in regime II](image-url)
A contour plot is preformed for the abnormal condition of 3 fans fail and 6 fans fail at the time of maximum temperature values and depicted in Figs. 5.24 and 5.25 for regimes I and II respectively. It is found that, the maximum temperature of clad surface in regime I is (80.5°C), and it occurs at the 58.5 cm height above core entrance. While for regime II, the maximum clad surface temperature is (104.6°C), and it occurs at the 58.5 cm height above core entrance.
Fig. (5.25) Temperature contour of clad surface in abnormal regime II.
CHAPTER (6)

CONCLUSION AND RECOMMENDATION

6.1 Conclusion

A thermal-hydraulic model is formulated using finite difference approximation of a group of differential equations that describe the reactor core, heat exchanger, pipe lines and cooling tower. The model describes the reactor cooling system and predicts the fuel, clad, coolant maximum temperatures in the core, heat exchangers and cooling tower in steady and transient state. The model is validated by PARET code for steady-state operation and validated also by the reactor operation records for transients in normal operating condition. The model is used to simulate the loss of the ultimate heat sink accident described by one through five cells failure in addition to complete failure of heat sink.

Steady state results show that the maximum temperatures of fuel, clad and coolant outlet temperature in both average channel and hot channel are 75.0°C, 65.5°C, and 50°C for average channels respectively, and 117.0°C, 97.5°C, 63°C for hot channel respectively. The model predicts the various maximum temperatures of fuel clad and coolant outlet temperatures during the transient in case of loss of heat sink to predict these values and the time that it takes place to trigger the safety action accompanied by the reactor protection system (RPS), two types of these safeties are used for triggering:

1- Power reduction safety action.
2- Scram emergency shutdown.

The temperatures are calculated based on operating the reactor in both regimes I (11MW) and regime II (22MW) to differentiate between them to know the necessary time for reactor protection system to trigger the safety.
In case of regime I, the fuel, clad and outlet coolant temperatures are respectively 87.8°C, 80.4°C and 64.7°C in hot channel while the power reduction safety is triggered at a time 680sec. These values are changed to 88.9°C, 81.8°C, and 65.8°C in case of emergency shutdown to take place a triggering in a times 570sec. and 450sec. with two and three fans failure respectively. In case of regime II, the fuel, clad and outlet coolant temperatures are respectively 119.2°C, 102.6°C and 66.75°C in hot channel and stabilizes at these values with safe operation after a time 1500sec., while these values are changed to 119.9°C, 102.8°C, and 67.5°C in case of power reduction safety triggering emergency to be taken place, on the other hand a scram safety shut down is triggering at a times 600 sec., 440 sec., 310 sec. and 240 sec. in cases of failure of three, four, five and six fans respectively while the predicted temperatures of fuel, clad and outlet coolant are predicted 120.8°C, 104°C and 68.6°C respectively. The maximum clad temperature are compared with the heat transfer limits, The maximum onset of nucleate boiling is calculated and equals (127.2°C), the saturated temperature is considered (117°C), Furthermore, the onset of nucleate boiling and departure from nucleate boiling are investigated on the corresponding heat fluxes, the safety margins are calculated in hot channels of both of regime (I) and (II), the result shows a good and satisfactory margin in three aforementioned cases.
6.2 Recommendation of future work

Based on the present model, it can be developed as follows:

1- The present model can be modified to include the reactivity effect by implementing a model for point kinetic equations.

2- The present model can be modified to simulate the reactor cooling system performance during loss of flow accident (LOFA).
REFERENCES


3-M. A. Raouf; H. Khater; and S El-Din El-Morshdy (Thermal hydraulic modeling of the onset of flow instability in MTR reactors) Thesis of doctoral, faculty of engineering, Cairo University, 2006.


8-S. H. Weiss (Guidelines for Preparing and Reviewing Applications for the Licensing of Non-Power Reactors) NUREG-1537, part 1, 1996.

9-S. G. Kandlikar (Heat transfer characteristics in partial boiling, fully developed boiling and significant void flow regions of subcooled flow boiling) Journal of heat transfer; Vol. 120/295 1998.


12-S. El-Din El-Morshedy (Predictive study of the onset of flow instability in narrow vertical rectangular channels under low pressure sub-cooled boiling) Nuclear Engineering and Design 244, Pages 34–42, 2012.


21-A. Salama and S. El-Din El-Morshedy (3D thermal hydraulic simulation of the hot channel of a typical material testing reactor under normal operation conditions) Kerntechnik 75; 2010.


27-Sofu and Dodds (A computer model for the transient analysis of compact research reactors with plate type fuel) International topical meeting on the safety of advanced reactors, Pittsburgh, pages18-20, 1994.


31-J. Baggemann, St. Kelm, Herve´ Guyon, J. Lehmkuhl and Hans-Josef Allelein (CFD Modeling of the Thermal Hydraulics inside a Spent Fuel


44-W. Maximo Torres, Benedito Dias Baptista Filho and Daniel Kao Sun Ting (Development of an emergency core cooling system for the converted IEA-R1M research reactor) International Meeting on Reduced Enrichment for Research and Test Reactors, São Paulo, Brazil, pages 18-23m 1998.


52. S. El-Din El Morshedy (Determination of the replacement cooling tower capability at the ETRR-2 research reactor) KERNTECHNIK; 69; 2004.


66-D. G KrÖger "Air cooled heat exchangers and cooling towers" PennWell Corp; Tulsa, Oklahoma; 2004.


APPENDIX (1)

Program model in EES
Function abc(T_i_average, T_co)
    if (T_i_average >= 43.2) and (T_co > 54.2) then CALL ERROR(' T_i_average must be less than 43.')
    abc := T_i_average
end

"Hot Channel"

Subprogram FuelTemp_h(q_a, PPF_h, U, A_f, A_cl, M_f, c_pf, T_f_h, T_cl_h: dt_f_h)
    dt_f_h = (q_a * PPF_h * A_f) / (M_f * c_pf) - ((U' * (A_f + A_cl)) / (M_f * c_pf)) * (T_f_h - T_cl_h)
END

Subprogram CladTemp_h(U, A_f, T_f_h, T_cl_h, h, A_cl, M_f, c_pf, M_cl, c_pclh, T_co_h: dt_cl_h)
    dT_cl_h = ((U' * (A_f + A_cl)) / (M_f * c_pf)) * (T_f_h - T_cl_h) -
              ((h * A_cl) / (M_cl * c_pclh)) * (T_cl_h - T_co_h)
END

Subprogram CoolantTemp_h(flow, T_i_h, h, A_cl, T_cl_h, T_co_h, Mco, c_p: dt_c_h)
    dt_c_h = -flow * (T_co_h - T_i_h) / (Mco) + h * A_cl * (T_cl_h - T_co_h) / (Mco * c_p)
END

Procedure RKCH(Time, T_cl_h0, T_c_h0: T_co_h)
    stept = 0.001
    t := 1
    T_co_h := T_c_h0

    :10
    IF(t < Time) THEN
        Call CoolantTemp_h(flow, T_i_h, h, A_cl, T_cl_h, T_co_h, Mco, c_p: dt_c_h)
q1 = stept * dt_c_h
T_co1 = T_co_h + q1 / 2
Call CoolantTemp_h(flow, T_i_h, h, A_cl, T_cl_h, T_co1, Mco, c_p: dt_c_h)
q2 = stept * dt_c_h
T_co2 = T_co1 + q2 / 2
Call CoolantTemp_h(flow, T_i_h, h, A_cl, T_cl_h, T_co2, Mco, c_p: dt_c_h)
q3 = stept * dt_c_h
T_co3 = T_co2 + q3 / 2
q4 = stept * dt_c_h
T_co_h = T_co_h + 1/6*(q1+2*q2+2*q3+q4)
t = t + 1
GOTO 10
ENDIF
END RKC

"*******************************************************************************
Procedure RK1_h(Time, T_f_h0, T_cl_h0, T_c_h0: T_f_h, T_cl_h)
$Common
flow, T_i_h, h, M_cl, c_pclh, U, A_f, M_f, PPF_h, A_cl, c_pf, Mco, c_p, S, N_fe, N_fp, q_a, T_co_h
step = 0.001
T_f_h := T_f_h0
T_cl_h := T_cl_h0
stept = 0.0001
t := 1
:10
IF(t<Time) THEN
Call FuelTemp_h(q_a, PPF_h, U, A_f, A_cl, M_f, c_pf, T_f_h, T_cl_h: dt_f_h)
Call CladTemp_h(U, A_f, T_f_h, h, A_cl, M_f, c_pf, M_cl, c_pclh, T_cl_h, T_co_h: dt_cl_h)
q1 = stept * dt_f_h
r1 = stept * dt_cl_h
T_f1 = T_f_h + q1 / 2
T_cl1 = T_cl_h + r1 / 2
Call FuelTemp_h(q_a, PPF_h, U, A_f, A_cl, M_f, c_pf, T_f1, T_cl1: dt_f_h)
Call CladTemp_h(U,A_f,T_f1,h,A_cl,M_f,c_pf,M_cl,c_pclh,T_cl1,T_co_h:dt_cl_h)
q2=stept*dt_f_h
r2=stept*dt_cl_h
T_f2=T_f1+q2/2
T_cl2=T_cl1+r2/2
Call FuelTemp_h(q_a,PPF_h,U,A_f,A_cl,M_f,c_pf,T_f2,T_cl2:dt_f_h)
Call CladTemp_h(U,A_f,T_f2,h,A_cl,M_f,c_pf,M_cl,c_pclh,T_cl2,T_co_h:dt_cl_h)
q3=stept*dt_f_h
r3=stept*dt_cl_h
T_f3=T_f2+q3/2
T_cl3=T_cl2+r3/2
Call FuelTemp_h(q_a,PPF_h,U,A_f,A_cl,M_f,c_pf,T_f3,T_cl3:dt_f_h)
Call CladTemp_h(U,A_f,T_f3,h,A_cl,M_f,c_pf,M_cl,c_pclh,T_cl3,T_co_h:dt_cl_h)
q4=stept*dt_f_h
r4=stept*dt_cl_h
T_f_h=T_f_h+1/6*(q1+2*q2+2*q3+q4)
T_cl_h=T_cl_h+1/6*(r1+2*r2+2*r3+r4)
t=t+1
GOTO 10
ENDIF
END RK1_h

"**************************************************************************************************************
Subprogram FuelTemp(q_a,PPF,U,A_f,A_cl,M_f,c_pf,T_f,T_cl:dt_f)
dt_f=(q_a*PPF*(A_f))/(M_f*c_pf)-((U*(A_f+A_cl))/(M_f*c_pf))*(T_f-T_cl)
END

**************************************************************************************************************
Subprogram CladTemp(U,A_f,T_f,h,A_cl,M_f,c_pf,M_cl,c_pcl,T_cl,T_co:dt_cl)
$Common T_co
dT_cl=((U*(A_f+A_cl))/(M_f*c_pf))*(T_f-T_cl)-((h*A_cl)/(M_cl*c_pcl))*(T_cl-T_co)
END

**************************************************************************************************************"
Subprogram CoolantTemp(flow,T_i,h,A_cl,T_cl,T_co,Mco,c_p:dt_c)

\[
dt_c = \frac{-\text{flow} \times (T_{co} - T_i)}{(Mco)} + h \times A_{cl} \times \frac{(T_{cl} - T_{co})}{(Mco \times c_p)}
\]

END

"******************************************************************************"  
Procedure RKCA(Time,T_cl0,T_c0:T_co)

$Common flow,T_i,h,A_cl,Mco,c_p,T_cl,T_co$

stept=0.001

t:=1

T_co:=T_c0

10:  
IF(t<Time) THEN
  Call CoolantTemp(flow,T_i,h,A_cl,T_cl,T_co,Mco,c_p:dt_c)
  q1=stept*dt_c
  T_co1=T_co+q1/2
  Call CoolantTemp(flow,T_i,h,A_cl,T_cl,T_co1,Mco,c_p:dt_c)
  q2=stept*dt_c
  T_co2=T_co1+q2/2
  Call CoolantTemp(flow,T_i,h,A_cl,T_cl,T_co2,Mco,c_p:dt_c)
  q3=stept*dt_c
  T_co3=T_co2+q3/2
  q4=stept*dt_c
  T_co=T_co+1/6*(q1+2*q2+2*q3+q4)
  t=t+1
  GOTO 10
ENDIF

END RKCA

"******************************************************************************"  
Procedure RK1(Time,T_f0,T_cl0,T_c0:T_f,T_cl)

$Common flow,T_i,h,A_cl,T_cl,M_cl,c_pcl,U,A_f,M_f,PPF,A_cl,c_pf,Mco,c_p,S,N_fe,N_fp,q_a,T_co$

step=0.001

T_f:=T_f0
T_cl=T_cl0
stept=0.0001
t:=1
:10
IF(t<Time) THEN
Call FuelTemp(q_a,PPF,U,A_f,A_cl,M_f,c_pf,T_f,T_cl:dt_f)
Call CladTemp(U,A_f,T_f,h,A_cl,M_f,c_pf,M_cl,c_pcl,T_cl,T_co:dt_cl)
q1=stept*dt_f
r1=stept*dt_cl
T_f1=T_f+q1/2
T_cl1=T_cl+r1/2
Call FuelTemp(q_a,PPF,U,A_f,A_cl,M_f,c_pf,T_f1,T_cl1:dt_f)
Call CladTemp(U,A_f,T_f1,h,A_cl,M_f,c_pf,M_cl,c_pcl,T_cl1,T_co:dt_cl)
q2=stept*dt_f
r2=stept*dt_cl
T_f2=T_f1+q2/2
T_cl2=T_cl1+r2/2
Call FuelTemp(q_a,PPF,U,A_f,A_cl,M_f,c_pf,T_f2,T_cl2:dt_f)
Call CladTemp(U,A_f,T_f2,h,A_cl,M_f,c_pf,M_cl,c_pcl,T_cl2,T_co:dt_cl)
q3=stept*dt_f
r3=stept*dt_cl
T_f3=T_f2+q3/2
T_cl3=T_cl2+r3/2
Call FuelTemp(q_a,PPF,U,A_f,A_cl,M_f,c_pf,T_f3,T_cl3:dt_f)
Call CladTemp(U,A_f,T_f3,h,A_cl,M_f,c_pf,M_cl,c_pcl,T_cl3,T_co:dt_cl)
q4=stept*dt_f
r4=stept*dt_cl
T_f=T_f+1/6*(q1+2*q2+2*q3+q4
T_cl=T_cl+1/6*(r1+2*r2+2*r3+r4

GOTO 10
ENDIF
END RK
Subprogram pipecorehx(A, Vel, Rho, m_cv, T_co, T_p_chx: dt_p_chx)

\[ dt_p_chx = \left(\frac{A \cdot Vel \cdot Rho}{m_cv}\right) \cdot \left(T_co - T_p_chx\right) \]

END

Procedure RK2(Time, T_co, T_p_chx0: T_p_chx)

$Common A, Vel, Rho, m_cv$

T_p_chx := T_p_chx0

stept = 0.5

t := 1

:10

IF(t < Time) THEN

Call pipecorehx(A, Vel, Rho, m_cv, T_co, T_p_chx: dt_p_chx)

q1 = stept*dt_p_chx

T_p_chx1 = T_p_chx + q1/2

Call pipecorehx(A, Vel, Rho, m_cv, T_co, T_p_chx1: dt_p_chx)

q2 = stept*dt_p_chx

T_p_chx2 = T_p_chx1 + q2/2

Call pipecorehx(A, Vel, Rho, m_cv, T_co, T_p_chx2: dt_p_chx)

q3 = stept*dt_p_chx

T_p_chx3 = T_p_chx2 + q3/2

Call pipecorehx(A, Vel, Rho, m_cv, T_co, T_p_chx3: dt_p_chx)

q4 = stept*dt_p_chx

T_p_chx = T_p_chx + 1/6*(q1 + 2*q2 + 2*q3 + q4)

t = t + 1

GOTO 10

ENDIF

END RK

Subprogram

HXSO1(U_hx1, A_hx, w_s, c_pc, T_hrxo1, T_rxo1, T_rhl1, T_feed, m_s: dt_hrxo1)

\[ DIV = \left(\frac{T_rhl1 - T_hrxo1}{T_rxo1 - T_feed}\right) \]
DELTAT_lm1=((T_rhl1-T_hrxo1)-(T_rxo1-T_feed))/ln(DIV)
dt_hrxo1=(U_hx1*A hx*DELTAT_lm1-w_s*c_pc*(T_hrxo1-T_feed))/(m_s*c_pc)
END

"******************************************************************************************
Program
HXPO1(w_p,c_pc,T_rhl1,T_rxo1,T_hrxo1,T_feed,U_hx1,A_hx,m_p:dt_rxo1)
DIV=((T_rhl1-T_hrxo1)/(T_rxo1-T_feed))
DELTAT_lm=((T_rhl1-T_hrxo1)-(T_rxo1-T_feed))/ln(DIV)
dT_rxo1=(w_p*c_pc*(T_rhl1-T_rxo1)-U_hx1*A hx*DELTAT_lm)/(m_p*c_pc)
END

"******************************************************************************************
Procedure RKHX1(Time,T_hrx0,T_rx0:T_hrxo1,T_rxo1)
$Common w_p,T_rhl1,m_p,c_pc,m_s,w_s,T_feed,U_hx1,A_hx
T_hrx0:=T_hrx0
T_rx0=T_rx0
stept=0.01
t=1
:10
IF(t<Time) THEN
Call HXSO1(U_hx1,A_hx,w_s,c_pc,T_hrxo1,T_rxo1,T_rhl1,T_feed,m_s:dt_hrxo1)
Call HXPO1(w_p,c_pc,T_rhl1,T_rxo1,T_hrxo1,T_feed,U_hx1,A_hx,m_p:dt_rxo1)
q1=stept*dt_hrxo1
r1=stept*dt_rxo1
T_hrxo11=T_hrxo1+q1/2
T_rxo11=T_rxo1+r1/2
Call HXSO1(U_hx1,A_hx,w_s,c_pc,T_hrxo11,T_rxo11,T_rhl1,T_feed,m_s:dt_hrxo1)
Call HXPO1(w_p,c_pc,T_rhl1,T_rxo11,T_hrxo11,T_feed,U_hx1,A_hx,m_p:dt_rxo1)
q2=stept*dt_hrxo1
r2=stept*dt_rxo1
T_hrxo12=T_hrxo11+q2/2
T_rxo12=T_rxo11+r2/2
Call HXSO1(U_hx1,A_hx,w_s,c_pc,T_hrxo12,T_rxo12,T_rhl1,T_feed,m_s:dt_hrxo1)
Call HXPO1(w_p,c_pc,T_rhl1,T_rxo12,T_hrxo12,T_feed,U_hx1,A_hx,m_p:dt_rxo1)
q3=stept*dt_hrxo1

r3=stept*dt_rxo1
T_hrxo13=T_hrxo12+q3/2
T_rxo13=T_rxo12+r3/2
Call HXSO1(U_hx1,A_hx,w_s,c_pc,T_hrxo13,T_rxo13,T_rhl1,T_feed,m_s:dt_hrxo1)
Call HXPO1(w_p,c_pc,T_rhl1,T_rxo13,T_hrxo13,T_feed,U_hx1,A_hx,m_p:dt_rxo1)
q4=stept*dt_hrxo1
r4=stept*dt_rxo1
T_hrxo1=T_hrxo1+1/6*(q1+2*q2+2*q3+q4)
T_rxo1=T_rxo1+1/6*(r1+2*r2+2*r3+r4)
t=t+1
GOTO 10
ENDIF
END RKHX1

"******************************************************************************
Subprogram
HXSO2(U_hx2,A_hx,w_s,c_pc,T_hrxo2,T_rxo2,T_rhl2,T_feed,m_s:dt_hrxo2)
DIV=((T_rhl2-T_hrxo2)/(T_rxo2-T_feed))
DELTAT_lm2=((T_rhl2-T_hrxo2)-(T_rxo2-T_feed))/ln(DIV)
dt_hrxo2=(U_hx2*A_hx*DELTAT_lm2-w_s*c_pc*(T_hrxo2-T_feed))/(m_s*c_pc)
END
"******************************************************************************
Subprogram
HXPO2(w_p,c_pc,T_rhl2,T_rxo2,T_hrxo2,T_feed,U_hx2,A_hx,m_p:dt_rxo2)
DIV=((T_rhl2-T_hrxo2)/(T_rxo2-T_feed))
DELTAT_lm=((T_rhl2-T_hrxo2)-(T_rxo2-T_feed))/ln(DIV)
dT_rxo2=(w_p*c_pc*(T_rhl2-T_rxo2)-U_hx2*A_hx*DELTAT_lm)/(m_p*c_pc(}
END
"******************************************************************************
Procedure RKHX2(Time,T_hrx00,T_rx00:T_hrxo2,T_rxo2)
SCommon w_p,T_rhl2,m_p,c_pc,m_s,w_s,T_feed,U_hx2,A_hx
T_hrxo2:=T_hrx00
T_rxo2=T_rx00
stept=0.01
t:=1
:10
IF(t<Time) THEN
Call HXSO2(U_hx2,A_hx,w_s,c_pc,T_hrxo2,T_rxo2,T_rhl2,T_feed,m_s:dt_hrxo2)
Call HXPO2(w_p,c_pc,T_rhl2,T_rxo2,T_hrxo2,T_feed,U_hx2,A_hx,m_p:dt_rxo2)
q1=stept*dt_hrxo2
r1=stept*dt_rxo2
T_hrxo21=T_hrxo2+q1/2
T_rxo21=T_rxo2+r1/2
Call HXSO2(U_hx2,A_hx,w_s,c_pc,T_hrxo21,T_rxo21,T_rhl2,T_feed,m_s:dt_hrxo)
Call HXPO2(w_p,c_pc,T_rhl2,T_rxo21,T_hrxo21,T_feed,U_hx2,A_hx,m_p:dt_rxo2)
q2=stept*dt_hrxo2
r2=stept*dt_rxo2
T_hrxo22=T_hrxo21+q2/2
T_rxo22=T_rxo21+r2/2
Call HXSO2(U_hx2,A_hx,w_s,c_pc,T_hrxo22,T_rxo22,T_rhl2,T_feed,m_s:dt_hrxo2)
Call HXPO2(w_p,c_pc,T_rhl2,T_rxo22,T_hrxo22,T_feed,U_hx2,A_hx,m_p:dt_rxo2)
q3=stept*dt_hrxo2
r3=stept*dt_rxo2
T_hrxo23=T_hrxo22+q3/2
T_rxo23=T_rxo22+r3/2
Call HXSO2(U_hx2,A_hx,w_s,c_pc,T_hrxo23,T_rxo23,T_rhl2,T_feed,m_s:dt_hrxo2)
Call HXPO2(w_p,c_pc,T_rhl2,T_rxo23,T_hrxo23,T_feed,U_hx2,A_hx,m_p:dt_rxo2)
q4=stept*dt_hrxo2
r4=stept*dt_rxo2
T_hrxo2=T_hrxo2+1/6*(q1+2*q2+2*q3+q4
T_rxo2=T_rxo2+1/6*(r1+2*r2+2*r3+r4

ENDIF
END RK

******************************************************************

Subprogram pipecorehx1(A1,Vel1,Rho,m_cv1,Ti1,T_p_chx1o:dt_p_chxo1)
\[
dt_{p\_chx1}=(A_1*\text{Vel}_1*\text{Rho}/\text{m}_c\text{v}_1)*(\text{T}_i1-\text{T}_{p\_chx1o})
\]

END

"************************************************************************************************************"
Subprogram pipecorehx2(A_1,\text{Vel}_1,\text{Rho},\text{m}_c\text{v}_1,\text{T}_i2,\text{T}_{p\_chx2o}:\text{dt}_p\text{chxo2})
\[
dt_{p\_chx2o}=(A_1*\text{Vel}_1*\text{Rho}/\text{m}_c\text{v}_1)*(\text{T}_i2-\text{T}_{p\_chx2o})
\]
END

"************************************************************************************************************"
Procedure RKK(Time,\text{T}_i1,\text{T}_i2,\text{T}_{p\_chx1o0},\text{T}_{p\_chx2o0}:\text{T}_{p\_chx1o},\text{T}_{p\_chx2o})
$Common A_1,\text{Vel}_1,\text{Rho},\text{m}_c\text{v}_1$
\[
\text{T}_{p\_chx1o}:=\text{T}_{p\_chx1o0}
\]
\[
\text{T}_{p\_chx2o}:=\text{T}_{p\_chx2o0}
\]
stept=0.5
\[
t:=1
\]
:10
IF(t<Time) THEN
\[
\text{Call pipecorehx1(A}_1,\text{Vel}_1,\text{Rho},\text{m}_c\text{v}_1,\text{T}_i1,\text{T}_{p\_chx1o}:\text{dt}_p\text{chxo1})
\]
\[
\text{Call pipecorehx2(A}_1,\text{Vel}_1,\text{Rho},\text{m}_c\text{v}_1,\text{T}_i2,\text{T}_{p\_chx2o}:\text{dt}_p\text{chxo2})
\]
\[
q_{11}=$stept*\text{dt}_p\text{chxo1}$
\]
\[
q_{21}=$stept*\text{dt}_p\text{chxo2}$
\]
\[
\text{T}_{p\_chx1o1}:=\text{T}_{p\_chx1o}+q_{11}/2
\]
\[
\text{T}_{p\_chx2o1}:=\text{T}_{p\_chx2o}+q_{21}/2
\]
\[
\text{Call pipecorehx1(A}_1,\text{Vel}_1,\text{Rho},\text{m}_c\text{v}_1,\text{T}_i1,\text{T}_{p\_chx1o1}:\text{dt}_p\text{chxo1})
\]
\[
\text{Call pipecorehx2(A}_1,\text{Vel}_1,\text{Rho},\text{m}_c\text{v}_1,\text{T}_i2,\text{T}_{p\_chx2o1}:\text{dt}_p\text{chxo2})
\]
\[
q_{12}=$stept*\text{dt}_p\text{chxo1}$
\]
\[
q_{22}=$stept*\text{dt}_p\text{chxo2}$
\]
\[
\text{T}_{p\_chx1o2}:=\text{T}_{p\_chx1o1}+q_{12}/2
\]
\[
\text{T}_{p\_chx2o2}:=\text{T}_{p\_chx2o1}+q_{22}/2
\]
\[
\text{Call pipecorehx1(A}_1,\text{Vel}_1,\text{Rho},\text{m}_c\text{v}_1,\text{T}_i1,\text{T}_{p\_chx1o2}:\text{dt}_p\text{chxo1})
\]
\[
\text{Call pipecorehx2(A}_1,\text{Vel}_1,\text{Rho},\text{m}_c\text{v}_1,\text{T}_i2,\text{T}_{p\_chx2o2}:\text{dt}_p\text{chxo2})
\]
\[
q_{13}=$stept*\text{dt}_p\text{chxo1}$
\]
\[
q_{23}=$stept*\text{dt}_p\text{chxo2}$
\]
\[
\text{T}_{p\_chx1o3}:=\text{T}_{p\_chx2o2}+q_{13}/2
\]
\[ T_{p\_chx2\_o3} = T_{p\_chx2\_o2} + \frac{q23}{2} \]

Call pipecorehx1(A1, Vel1, Rho, m_cv1, Ti1, T_{p\_chx1\_o3}: dt_{p\_chx1\_o1})

Call pipecorehx2(A1, Vel1, Rho, m_cv1, Ti2, T_{p\_chx2\_o3}: dt_{p\_chx2\_o2})

\[ q_{14} = \text{stept} \times dt_{p\_chx1\_o1} \]

\[ q_{24} = \text{stept} \times dt_{p\_chx2\_o2} \]

\[ T_{p\_chx1\_o} = T_{p\_chx1\_o} + \frac{1}{6}(q_{11} + 2q_{12} + 2q_{13} + q_{14}) \]

\[ T_{p\_chx2\_o} = T_{p\_chx2\_o} + \frac{1}{6}(q_{21} + 2q_{22} + 2q_{23} + q_{24}) \]

\[ t = t + 1 \]

GOTO 10

ENDIF

END RKK

"************************************************************************"

Subprogram CT(m_cv, m_dot_w, T_wi, T_w, m_dot_air, cp_w, h_ae, h_ai: dt_w)

\[ dt_w = \left( \frac{1}{m_cv} \right) \times \left( (m\_dot_w \times (T_wi - T_w)) - \left( \frac{m\_dot_air}{cp_w} \right) \times (h_ae - h_ai) \right) \]

END

"************************************************************************"

Procedure RKCT(Time, T_w0: T_w)

SCommon m_ctv, m_dot_w, m_dot_air, cp_w, h_ae, h_ai, T_i_average, T_hrxo1, T_w

\[ t_w = t_w0 \]

\[ hh = 0 \]

\[ c = 0 \]

\[ zzz = 0 \]

10:

IF(zzz <= Time) THEN

Call CT(m_ctv, m_dot_w, T_wi, T_w, m_dot_air, cp_w, h_ae, h_ai: dt_w)

\[ q1 = hh \times dt_w \]

\[ t_w1 = t_w + q1/2 \]

Call CT(m_ctv, m_dot_w, T_wi, T_w1, m_dot_air, cp_w, h_ae, h_ai: dt_w)

\[ q2 = hh \times dt_w \]

\[ t_w2 = t_w + q2/2 \]

Call CT(m_ctv, m_dot_w, T_wi, T_w2, m_dot_air, cp_w, h_ae, h_ai: dt_w)

\[ q3 = hh \times dt_w \]

155
t_w3 = t_w + q3/2
Call CT(m_ctv, m_dot_w, T_wi, T_w3, m_dot_air, cp_w, h_ae, h_ai; dt_w)
q4 = hh*dt_w
T_w = T_w + 1/6*(q1 + 2*q2 + 2*q3 + q4)
c = c + 1
hh = 0.1
zzz = hh*c
GOTO 10
ENDIF
END RK

Subprogram PTHX(A_s, Vel_s, Rho, m_cvs, T_is, T_s_hx; dt_s_hxo)
dt_s_hxo = ((A_s*Vel_s*Rho)/m_cvs)*(T_is - T_s_hx)
END

Procedure RKPT(Time, T_is, T_s_hx0; T_s_hx)
$Common A_s, Vel_s, Rho, m_cvs, T_i_average
T_s_hx := T_s_hx0
stept = 0.01
t := 1

:10
IF(t < Time) THEN
Call PTHX(A_s, Vel_s, Rho, m_cvs, T_is, T_s_hx; dt_s_hxo)
q1 = stept*dt_s_hxo
T_s_hx1 = T_s_hx + q1/2
Call PTHX(A_s, Vel_s, Rho, m_cvs, T_is, T_s_hx1; dt_s_hxo)
q2 = stept*dt_s_hxo
T_s_hx2 = T_s_hx1 + q2/2
Call PTHX(A_s, Vel_s, Rho, m_cvs, T_is, T_s_hx2; dt_s_hxo)
q3 = stept*dt_s_hxo
T_s_hx3 = T_s_hx2 + q3/2
Call PTHX(A_s, Vel_s, Rho, m_cvs, T_is, T_s_hx3; dt_s_hxo)
q4=stept*dt_s_hxo
T_s_hx=T_s_hx+1/6*(q1+2*q2+2*q3+q4)
t=t+1
GOTO 10
ENDIF
END RKPT
**********************************************************************
PT_safety=43
q_a=(Power)/(S*N_fe*N_fp)
T_i=T_i_average
T_i_h=T_i_average
H_e=.8
PPF=1.35
PPF_h=3
M_f=8500*19*29*.8*.064*0.0007
h=25274"(calculated from steady state analysis)
r_al=0.0004
k_n=13.6
k_al=157.8
n=0.00035"m"
rho_al=2.7E6/1000
M_cl=rho_al*r_al*.8*.064*2*19*29
Mco=94.483
S=0.1024
A_f=S*N_fe*N_fp
A_cl=.8*.07*2*N_fe*N_fp
c_pf=371
c_pclh="878"898
c_pcl=778
c_p=4182
1//U_f=n/(k_n)
1//U_cl=r_al/(k_al)
1/U=1/U_f+1/U_cl
m_dot=0.8978
N_fp=29
N_fe=19
m_dot_water=(m_dot/2)*N_fp*(N_fe+1)
Rho=996
m_dot_p=1950/2
flow=m_dot_p*Rho/3600
T_f0="60.816997"TableValue('11/1/-run',88,5)
T_cl0="53.099349"TableValue('11/1/-run',88,6)
T_c0="43.124062"TableValue('11/1/-run',88,7)
T_f_h0="T_f0"TableValue('11/1/-run',88,2)
T_cl_h0="T_cl0"TableValue('11/1/-run',88,3)
T_c_h0="T_c0"TableValue('11/1/-run',88,4)
Call RK1(Time,T_f0,T_cl0,T_c0:T_f,T_cl)
Call RK1_h(Time,T_f_h0,T_cl_h0,T_c_h0:T_f_h,T_cl_h)
Call RKCH(Time,T_cl_h0,T_c_h0:T_co_h)
Call RKCA(Time,T_cl0,T_c0:T_co)

"****************************************************************

"INPUT DATA OF PIPE FROM CORE TO HX"
AB=11.25+0.5"elbows""m"
D_AB=20*25.4/1000 "m"
R_1=1+0.010"m"
D_R1=25.4/1000 "m"
BC=11.5"m"
D_BC=14*25.4/1000 "m"
CD=1.5+0.2*6"fittings""m"
D_CD=14*25.4/1000 "m"
A=(Pi/4)*(D_AB^2+D_R1^2+D_BC^2+D_CD^2)/4
V=(Pi/4)*(D_AB^2*AB+2*R_1*D_R1^2+2*BC*D_BC^2+2*CD*D_CD^2)
Vel=3
m_cv=V*Rho
Ti=T_co
T_p_chx0="42.920858"TableValue('11/1/-run',88,8)
Call RK2(Time, Ti, T_p_chx0:T_p_chx)

"******************************************************************************

"INPUT DATA OF HEAT EXCHANGER NO. 1"

w_p=(1950/2)*996/3600
T_rhl1=T_p_chx

\(c_{pc} = 4200\)

m_p=506
m_s=538

w_s=(2700/2)*996/3600
T_feed=T_s_hx

\(U_{hx1} = 3440\)

A_hx=292.3

T_hrx0="33.043265"TableValue('11/1-run',88,10)
T_rx0="35.243265"TableValue('11/1-run',88,11)

Call RKHX1(Time, T_hrx0, T_rx0:T_hrxo1, T_rxo1)

"******************************************************************************

"INPUT DATA OF HEAT EXCHANGER NO. 2"

T_rhl2=T_p_chx

\(U_{hx2} = 3440\)

T_hrx00=33.173265"TableValue('11/1-run',88,9)
T_rx00=35.13265"TableValue('11/1-run',88,10)

"Call RKHX2(Time, T_hrx00, T_rx00:T_hrxo2, T_rxo2)

"******************************************************************************

DE=30+0.2*8+0.1*2+0.2*2+0.1 "m"

\(D_{DE} = 12*25.4/1000"m"\)

\(A1=(\pi/4)*(D_{DE}^2)\)

\(V1=(\pi/4)*(DE*D_{DE}^2)\)

\(Vel = 3\)

\(m_cv1 = V1*Rho\)

T_p_chx1o0="35.173265"TableValue('11/1-run',88,1)
T_p_chx2o0="35.073265"TableValue('11/1-run',88,16)
Ti1 = T_rxo1
Ti2 = T_rxo1
Call RKK(Time, Ti1, Ti2, T_p_chx1o0, T_p_chx2o0: T_p_chx1o, T_p_chx2o)

"******************************************************************************

T_i_average = (T_p_chx1o)

******************************************************************************

"INPUT DATA OF COOLING TOWER"
N_c = 6 "number of fan in cooling tower"
NOC_out = 5
cp_a = Cp(Air, T = T_db)
cp_w = 4.2
V_dot_air = 198400 * Convert(cf m, {m^3/s})
m_ctv = 14250 * (N_c - NOC_out) / (N_c) "kg"
Rho_water = Density(Water, T = 40, P = 101.03)
Rho_air = Density(Air, T = T_db, P = 101.03)
m_dot_w = (11657 / 2) * Convert(gal/min, {m^3/s}) * Rho_water
h_ai = Enthalpy(AirH2O, T = T_db, R = 0.44, P = 101.03) "inlet air Enthalpy"
DUPLICATE I = 1, (N_c - NOC_out)
m_dot[i] = V_dot_air * Rho_air
P[i] = 4200 "kw"
END
m_dot_air = sum(m_dot[i], I = 1, N_c - NOC_out)
P_o = sum(P[i], I = 1, N_c - NOC_out)
Ratio = m_dot_w / m_dot_air
P_o = m_dot_air * (h_ao - h_ai) "KW"
T_db = 34
Row = ABS(T_db * 0.5)
T = 2
P_tower = Po * 1000
T_wi = 37
T_w0 = "28.995489" TableValue('11/1/-run', 88, 13)
Call RKCT(Time, T_w0: T_w)
AF=2.5"16in"
BF=2"14' in"
CF=15.5"24in"
D_AF=16*25.4/1000"m"
D_BF=14*25.4/1000"m"
D_CF=24*25.4/1000"m"
Vs=(Pi/4)*(AF*D_AF^2+BF*D_BF^2+CF*D_CF^2)
m_cvs=Vs*Rho
A_s=(Pi/4)*(D_AF^2+D_BF^2+D_CF^2)/3
Vel_s=3
T_s_hx0="28.995489"TableValue('11/1-run',88,12)
T_is=T_w
Call RKPT(Time,T_is,T_s_hx0:T_s_hx)

Power=0.02E+6
"Call RPower(Time,Power_init:Power")
T_reg1=Time+390+870
g=abc(T_i_average,T_co)
T_in=44

"***********************************************************************

power=22E6 "Average thermal power of the reactor watt"
S=0.1024 "Heating surface of each fuel plate m^2"
A_thermal=0.4767E-4 "Cross section area of fuel core m^2"
N_fe=29
N_fp=19
m_dot=0.8078
e=0.0007 "m" "Thickness of the fuel plate"
R=3 "Flattening coefficient"
q_hot "Surface heat flux of hot channel"=117E4 "w/m^2"
c=0.40
c0=0.48

"***********************************************************************

"CORE THERMAL HYDRAULIC"
q_a=(power)/(S*N_fe*N_fp) "Average heat flux"=390121
Q_v=q_a/(e/2) "Average volumetric heat generation"
qmax=q_a*R
Q_max=Q_v*R
rho=Density(Water,T=T_mz,P=101.03)
c_p=Cp(Water,T=T_mz,P=301.03)*1000".99919*Convert(cal/g-J/kg-K")
Q_z"w/m^3"=Q_max"w/m^3"*cos((180*z)/(2*c0))
qz=qmax*cos((180*z)/(2*c0))
"dq_g heat generated per unit time=Q_z*A_thermal*dz"
"dq_e heat removed per unit time=m_dot*c_p*dT"
"dq_g=dq_e"

(m_dot*c_p)*(T_mz"Coolant tem. at axial z direction")-
T_i)=(Q_max*A_thermal)*((2*c/pi)*sin((180*z)/(2*c0))+sin((180*c)/(2*c0)))
T_mz0=TableValue('Table1',21,4)
A_cc=1.917E-4"m^2"
p=2*(7.1+0.27)*1E-2*0.1464m
nu=5.384E-7
m_dot=rho*A_cc*v "Average coolant velocity in z=0"
D_h=4*A_cc/p"m"
R_e=(D_h*v/nu)
k=Conductivity(Water,T=T_mz,P=101.03)
mu=Viscosity(Water,T=T_mz,P=101.03)
Nusselt=h*D_h/k
Nusselt=0.0243*(R_e)^0.8*(Pr)^0.4
Pr=mu*c_p/k
T pz0=T mz0+q_a*R/h
qz=h*(T pz-T mz)
qz=((2*n/k_n)+(2*r_al/k_al))^(-1)*(T mnz-T mz)
q_z6=TableValue('Table1',21,2)
q_x=q_z6
Tmep"maximum exterior temp. of cladding"=TableValue('Table1',24,5)
q_x=(k al/r al)*(T mir"maximum exterior temp. of cladding"-Tmep)
Qz6=TableValue('Table1',24,3)
k n=13.76"w/m^2"
x=0
n=0.00035"m"
*2)*k n)*(T mn"maximum temp of fuel core"-T mir)/(n^2-x^2)=Qz6
"******************************************"
"HEAT TRANSFER LIMIT AND SAFETY MARGIN"
bar=2
P_c=208
T_sat=117
h fg=2200
W=0.064
ETA=25
W_h=.046
D=0.0027
L=0.80
T_i=40
power=22E6"Average thermal power of the reactor watt"
S=0.1024 "Heating surface of each fuel plate m^2"
A_th=(0.4667E-4)/(1)"cross section area of fuel element m^2"
N_fe=29
N_fp=19
m_dot1=0.9981/2
m_dot2=0.9981/2
m_dot=0.9981
F=if(Power,11E6,m_dot1,m_dot2,m_dot)
R_h=3"flattening coefficient"
R_a=1.35
"F=if(Power,11E6,m_dot1,m_dot2,m_dot)
k_al=157.8
r_al=0.0004
k_n=13.8"w/m^2"
n=0.0007"m"
c=0.40
c=0.48
q_a=(power)/(S*N_fe*N_fp)"average heat flux"390121
Q_v=q_a/(n/2)"average volumetric heat generation"
qm_a=q_a*R_a
qm_h=q_a*R_h
Q_hh=Q_v*R_h
Q_aa=Q_v*R_a
rho=Density(Water,T=T_mzh,P=101.03(C_p=Cp(Water,T=T_mzh,P=101.03)*1000*.99919*Convert(cal/g-k.j/kg-K)
**********************************************************
qza=qm_a*cos((180*z)/(2*co))
qzh=qm_h*cos((180*z)/(2*co))
**********************************************************
Q_za"w/m^3"=Q_aa"w/m^3"*cos((180*z)/(2*co))
Q_zh"w/m^3"=Q_hh"w/m^3"*cos((180*z)/(2*co))
**********************************************************
T_mza-T_i=(Q_aa*A_th/(F*C_p))*(2*Co/pi)*(sin((180*z)/(2*co))-sin((180*(-c))/(2*co)))
qza=h*(T_pza-T_mza)
\[ T_{mzh} - T_i = \left( \frac{Q_{hh} A_{th}}{(F \cdot C_p)} \right) \cdot (2 \cdot \frac{Co}{\pi}) \cdot \left( \sin((180^{\circ} z)/(2 \cdot co)) - \sin((180^{\circ} (-c))/(2 \cdot co)) \right) \]

\[ qzh = h \cdot (T_{pzh} - T_{mzh}) \]

\[ -T_{mira} + T_{pza} = (-qza/k_al) \cdot (r_al) \]

\[ -T_{mirh} + T_{pzh} = (-qzh/k_al) \cdot (r_al) \]

\[ T_{mnza} = T_{mira} + \left( \frac{Q_{za}}{(2 \cdot k_n)} \right) \cdot \left( \frac{n}{2} \right)^2 \cdot 1.225 \times 10^{-7} \]

\[ T_{mnzh} = T_{mirh} + \left( \frac{Q_{zh}}{(2 \cdot k_n)} \right) \cdot \left( \frac{n}{2} \right)^2 \cdot 1.225 \times 10^{-7} \]

\[ A_{cc} = 1.917 \times 10^{-4} \text{m}^2 \]

\[ p = 2 \cdot (7.1 + 0.27) \times 10^{-2} \cdot 0.1464 \text{m} \]

\[ nu = 5.384 \times 10^{-7} \]

\[ F = \rho \cdot A_{cc} \cdot v \] average coolant velocity in z=0

\[ D_h = 4 \cdot A_{cc} / p \text{m} \]

\[ R_e = (D_h \cdot v / nu) \]

\[ k = \text{Conductivity(Water, T=T_{mzh}, P=101.03)} \]

\[ \mu = \text{Viscosity(Water, T=T_{mzh}, P=101.03)} \]

\[ "Nusselt = h \cdot D_h / k" \]

\[ h = 25213 \]

\[ \text{Nusselt} = 0.0243 \cdot (R_e)^{0.8} \cdot (Pr)^{0.4} \]

\[ Pr = \mu \cdot c_p / k \]

\[ T_{onb} = \Delta T_{sat} + T_{sat} \]

\[ var = (q_{ad} / (1082 \times \text{bar}^{1.156})) \]

\[ T_{onb} = 0.556 \cdot \text{var}^{0.463} \cdot \text{bar}^{0.0234} + T_{sat} \]

\[ \text{PHI}_{onb} = h \cdot (T_{onb} - 66 + T_{mzhm}) \]

\[ R_s = 1 / (1 + ETA \cdot (D_h / L)) \]

\[ \text{PHI}_{ofi} = (R_s \cdot c_p \cdot \text{RHO} \cdot V \cdot ((W \cdot D) / (W_h \cdot L)) \cdot (T_{sat} - 44 + T_i) \]

\[ \text{PHI}_{dnb} = 1454 \cdot \text{Theta} \cdot (((1 + (2.5 \cdot V^2) / \text{Theta}))^{0.25}) \cdot (1 + 15.1 \cdot C_p \cdot (\text{DELTAT}_{sub} / (h \_fg \cdot \text{bar} ^{0.5}))) \]

\[ \Theta = 0.99531 \cdot ((\text{bar}^{1/3}) \cdot (1 - (\text{bar} / P_c))^{4/3}) \]

\[ \text{DELTAT}_{sub} = T_{sat} - 66 + T_{mzhm} \]

165
Ratio_onb = PHI_onb / q_ad
Ratio_ofi = PHI_ofi / q_ad
Ratio_dnb = PHI_dnb / q_ad
T_mzhm = MaxParametric('Table 13', 'T_mzh')
q_ad = MaxParametric('Table 13', 'qm_h')
APPENDIX (2)
DECAY POWER
The normalized decay power after Scram is shown in table B.1, according to ANS tables. Based on this table, the normalized decay power is depicted in Fig. B.1 for 10 minutes.

Table B.1 Normalized power after Scram.

<table>
<thead>
<tr>
<th>Time after SCRAM (Seconds)</th>
<th>Normalized power</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>0.069900</td>
</tr>
<tr>
<td>1.0E-01</td>
<td>0.068900</td>
</tr>
<tr>
<td>1.0E+00</td>
<td>0.062800</td>
</tr>
<tr>
<td>2.0E+00</td>
<td>0.059200</td>
</tr>
<tr>
<td>4.0E+00</td>
<td>0.055400</td>
</tr>
<tr>
<td>6.0E+00</td>
<td>0.053100</td>
</tr>
<tr>
<td>8.0E+00</td>
<td>0.051400</td>
</tr>
<tr>
<td>1.0E+01</td>
<td>0.049900</td>
</tr>
<tr>
<td>2.0E+01</td>
<td>0.044900</td>
</tr>
<tr>
<td>4.0E+01</td>
<td>0.039800</td>
</tr>
<tr>
<td>6.0E+01</td>
<td>0.037200</td>
</tr>
<tr>
<td>8.0E+01</td>
<td>0.035300</td>
</tr>
<tr>
<td>1.0E+02</td>
<td>0.033700</td>
</tr>
<tr>
<td>2.0E+02</td>
<td>0.028400</td>
</tr>
<tr>
<td>4.0E+02</td>
<td>0.023400</td>
</tr>
<tr>
<td>6.0E+02</td>
<td>0.021300</td>
</tr>
<tr>
<td>8.0E+02</td>
<td>0.020100</td>
</tr>
<tr>
<td>1.0E+03</td>
<td>0.019200</td>
</tr>
<tr>
<td>2.0E+03</td>
<td>0.016100</td>
</tr>
<tr>
<td>4.0E+03</td>
<td>0.012800</td>
</tr>
<tr>
<td>6.0E+03</td>
<td>0.011200</td>
</tr>
<tr>
<td>8.0E+03</td>
<td>0.010300</td>
</tr>
<tr>
<td>1.0E+04</td>
<td>0.009760</td>
</tr>
<tr>
<td>2.0E+04</td>
<td>0.008010</td>
</tr>
<tr>
<td>4.0E+04</td>
<td>0.006260</td>
</tr>
<tr>
<td>6.0E+04</td>
<td>0.005460</td>
</tr>
<tr>
<td>8.0E+04</td>
<td>0.005050</td>
</tr>
<tr>
<td>1.0E+05</td>
<td>0.004790</td>
</tr>
<tr>
<td>2.0E+05</td>
<td>0.004090</td>
</tr>
<tr>
<td>4.0E+05</td>
<td>0.003390</td>
</tr>
</tbody>
</table>
Figure B.1 Normalized decay power after Scram.
وقد تناولت الرسالة أيضا بعض نتائج سطع غلاف وقود اليورانيوم المصنوع من الألومنيوم في الأتجاهين اتجاه (العمق والأرتفاع) في صورة كونتور يوضح درجات الحرارة عند أقصى موضع وكذلك درجات الحرارة المحيط بها.

الفصل السادس:

يحتوي على الاستنتاجات النهائية من هذا البحث وتقييم ما تم التوصل إليه من نموذج رياضي مقترح يحاكي هذه النوعية من مفاعلات الأبحاث وملائمته لمحاكاة مثل هذه الحوادث وتقييم أوضاع الأمان الحرارية فيها ومدى قربها أو بعدها عن الحدود الحرارية الحرجة التي يؤدي تعديها إلى قيود على القدرة التشغيلية للمفاعل كما تضمن التوصيات والمقترحات المستقبلية التي يمكن من خلالها أضافة تطورات أخرى تحاكي أنواع أخرى من هذه الحوادث.
الفصل الخامس:

يحتوي هذا الفصل على النتائج المستنيرة من النموذج الرياضي المبتكر بدءاً من نتائج مرحلة الاستقرار لاستنتاج درجات حرارة منطقة قلب المفاعل وحتى المبادل الحراري وبرج التبريد وقد أظهرت النتائج صحة وثقة في نتائج النموذج الرياضي المقترح.

وقد تم استعراض دراسة في البعدين وهما طول وعرض قنوات التبريد لقلب المفاعل ورسم الشكل الكئي لدرجات حرارة السطح المصنع من الألومنيوم الخاص بألواح قضبان اليوانيوم الملائمة لواء التبريد.

ثم استعرض في هذا الفصل نتائج عدم الاستقرار بدءاً من مرحلة تشغيل المفاعل عند قدرة معينة وانتقالنا بالقدرة إلى قيمة أكبر واستقرارها مع مقارنتها أيضاً بالنتائج المقروءة والمسجلة بواسطة مجمع البيانات في المفاعل للوقوف على صحة الموديل الرياضي. 

ويحتوي هذا الفصل أيضاً النتائج المستنيرة في حالة الحادثة المفتعلة بدءاً من خروج أحد مراوح برج التبريد عن التشغيل ومروراً بالترد إلى انهيار برج التبريد لصل في نهاية النتائج إليه خروج برج التبريد نهائياً من الخدمة مع حساب الزمن لازم لاتخاذ نظام أمان المفاعل الأزر بالوقف الطارئ للمفاعل مستنداً إلى درجة حرارة دخول الماء لقلب المفاعل وأيضاً درجة حرارة خروج ماء التبريد من قلب المفاعل.

تمت مقارنة أقصى درجات حرارة مستنيرة بالحدود القصوى التي يحدث عدها بداية الغليان قريب الأسطح الساخنة لمعرفة ملائمة حدود الأمان للمفاعل لتلك القيم وضمان الحفاظ على المفاعل من تداعيات اقترابه من تلك الحدود.
المتوسط نظرا لتبان وضع هذه القنوات بالنسبة للفيض الحراري المتغير، فبذلك وضعها المتغير مع ارتفاع قلب المفاعل والتي تم التعبير عنها بمعادلة جيب التمام الموضحة في بداية النموذج الرياضي.

كما يحتوي النموذج الرياضي على معادلات لحل معادلات قضبان اليورانيوم المغلف بالألومنيوم في الأتجاهين أتجاه الأرتفاع وأتجاه العرض وذلك للتبني بالكانتور الحراري على سطح هذا الغلاف.

يحتوي هذا الفصل أيضا على المعادلات التفاضلية المعبرة عن درجات الحرارة في المبادل الحراري مستغلين المتغيرات والثوابت التصميمية المستخدمة في تصميم هذا المبادل للتبني بدرجات حرارة خروج المائع من الجانب الساخن والجانب البارد.

أما بالنظر الي برج التبريد فقد أحتوى هذا الفصل على المعادلات التفاضلية لبرج التبريد وتغيرها مع الزمن مع مراعاة التعبير الدقيق لسريان الهواء بواسطة مراوح برج التبريد والتي تستطيع من خلالها التعبير عن فقدان مراوح التبريد كحادثة مصطنعه تحاكي الحادثة الحقيقية لفقدان هذه المرواح نتيجة انقطاع التيار عنها أو تعرضه للإلف المفاجئ نتيجة لعطل ما.

وفي نهاية الفصل استعرض الحدود الحرجة لكل من درجات الحرارة والفيض الحراري والتي يبدأ عنها الغليان لمقارنة أقصى درجات حرارة قلب المفاعل ومدى القرب منها أثناء تلك الحادثة.

الفصل الرابع:

في هذا الفصل يتم تقييم النموذج الرياضي المبكر حيث يحتوي على نتائج النموذج الرياضي بعد تطبيقها على مفاعل مصرب البصطي الثاني محل الدراسة مع مقارنات تلك النتائج المستندة مع نتائج أخرى مقررة من غرفة تحكم وجمع البيانات بالمفاعل وذلك في حالة المستقرة والحياة الأنتقالية كما يتضمن أيضا مقارنات نتائج الحالة المستقرة والحياة الأنتقالية كما يتضمن أيضا مقارنات نتائج الحالة
الفصل الثاني:

يحتوي هذا الفصل على الأبحاث السابقة لمفاعلات الأبحاث وأختيار المواد سواء كانت أبحاث نظرية مستندة إلى نماذج رياضية أو أبحاث عملية وقد تم تقسيم الدراسة الاستراتيجية إلى ثلاث بنود أساسية البند الأول يحتوي على الأبحاث والدراسات التي أهدفت بالحوادث المحتملة حدوثها في هذه النوعية من المفاعلات وقد تركز هذا البند من الدراسة الاستراتيجية على الحوادث الأكثر شيوعًا وهي فقد معدل التصرف وفقد سائل التبريد وكذلك حوادث التفاعل النووي. البند الثاني يحتوي على النمذجة والمحاكاة لمفاعلات الأبحاث في مجال الهيدروليكا الحرارية مع استعراض لأنواع الأكواد المختلفة التي تم بها عملية المحاكاة والنمذجة. أخيرًا يحتوي البند الثالث على بعض الدراسات السابقة لمكونين أساسيين لأي نظام تبريد وعلا المبادلات الحرارية وأبراج التبريد سواء كانت هذه الأبحاث ترتبط بتطبيقات نووية أم غير نووية.

الفصل الثالث:

يحتوي هذا الفصل على النموذج الرياضي والذي يحتوي على معادلات لحل نظام تبريد المفاعل ومكوناته وهي:

- قلب المفاعل.
- المبادل الحراري.
- برج التبريد.
- خطوط الأنابيب.

ويحتوي النموذج الرياضي المبتكر على معادلات لحل حالة الاستقرار وأيضاً الحالة الانتقالية المتتغيرة مع الزمن للتنبؤ بدرجات حرارة قضبان اليورانيوم واغلفة الألومنيوم التي تحميها وراء التبريد المحيطي به في كل من القنوات الساخنة التي تتعرض لأقصى ضغط حراري والقنوات المتوسطة التي تتعرض للفيض الحراري.
المبرمج عليها نظام حماية الفاعل والتي على أساسها يأخذ قراره سواء بالأيَفاق القصري أو تقليل القدرة.

وتتتَسم الرسالة إلى ستة فصول:

الفصل الأول:
يتكون الفصل الأول من مقدمة الرسالة والتي تستعرض وصف مختصر لمفاهيم الأبحاث وسرد لأنواع تلك المفاهيم واستخداماتها السلمية سواء في الأبحاث العلمية أو في أنتاج النظائر المشعة المستخدمة طبيا، مع وصف مستوفي لمفاعل مصر البحثي الثاني كمثال للدراسة والتطبيق حيث يتكون هذا المفاعل من دائرة تبريد أساسية (أوليه) لتبريد قلب المفاعل وتنقسم تلك الدائرة الأولية إلى دائريتين منفصلتين سعة كل دائرة 11 ميجاوات يتم تشغيل احدهما عند تشغيل المفاعل بنصف الحمل والمسمى بالرجل واحد وتعمل على كل دائرة مبادل حراري من النوع ذو الألواح. والدائرة الثانية هي دائرة التبريد الثانية والتي تعتمد في تبريدها على استخدام برج تبريد من النوع الجبري ذو التلامس المباشر والمفتاح للهواء الجوسي ويتكون هذا البرج من عدد 6 وحدات تبريدية منفصلة بسعة تبريدية تقدر بحوالي 4.2 ميجاوات لكل وحدة وبأجمالى سعة تبريدية 25 ميجاوات تقريبًا. ومدعا البرسومات تفصيلية لدائرة التبريد والمكونات التصميمية لمكوناتها كقلب المفاعل والمبادل الحراري وبرج التبريد والتي سوف تستخدم في مدخلات النموذج الرياضي المبتكر. وقد تم استعراض الحوادث المحتملة حدوثها في هذا النوع من المفاعلات مع شرح موجز لأسباب حدوثها كما تم استعراض الحدود الحرجة الحرارية التي قد تواجه تشغيل تلك المفاعلات والتي يتم عمل حسابها أثناء تصميم هذه الأنواع من المفاعلات مع التوضيح اللازم لأسباب حدوث تلك الظواهر الحرجة وتأثيرها وأثارها على التشغيل.
نستطيع من خلال النموذج الرياضي المبتكر حساب درجات حرارة نظام التبريد في الحالات التالية:

1. الحالة المستقرة لكل من درجات حرارة دخول وخروج الماء لقلب المفاعل وأقسيم درجات حرارة كل من قضبان اليورانيوم وغلاف الألومينيوم وكل من فنات الساخنة والوسطاء وأيضاً درجة حرارة دخول وخروج ماء التبريد للمبادلات الحرارية وبرج التبريد.

2. الحالة النابضة (الغير مستقرة) من بداية رفع القدرة وحتى حالة الاستقرار عند قدرة معينة مستهدفة مع التنظيم بدرجات الحرارة لقلب المفاعل والمبادل الحراري وبرج التبريد.

3. محاكاة أحد حوادث المفاعلات والتي تتلخص في عدم قدرة برج التبريد على امتصاص الحرارة المثلى والقدرة من قلب المفاعل عن طريق توقف مراوح برج التبريد تدريجياً بدأ من توقف مراوح واحدة ثم مروحتين إلى أن نصل إلى توقف برج التبريد كلية واختبار نظام الأمان للمفاعل بتحديد الزمن اللازم والذي يستغرقه نهدف إلى الأيقاف الطارئي القصري للمفاعل في مثل هذه الحالات المختلفة.

4. مقارنة درجات الحرارة القصوى المنتجة من الحادثة لقلب المفاعل والتي تتمثل في درجات حرارة كل من الوقود وغلاف الألومينيوم وسائل التبريد مع مقارنتها بالحدود الحرجة والتي تخص مرحلة مختلفة للعباس وهي مرحلة بداية الغليان النوعي ومرحلة بداية السريان الغير مستقرة ومرحلة ما بعد الغليان النوعي وحساب معاملات آمان المفاعل لحدود تلك المرحلة، وتتبيين من الحسابات عدم حدوث تلك المراحل أثناء فقدان أبراج التبريد وذلك نظراً لقياس النظام المستند على حماية المفاعل بأطفاء المفاعل تفادياً لذلك. كما تم حساب درجات حرارة ماء التبريد داخل قلب المفاعل وثبت بعدها عن القيم الحرج. لتمثل هذا النوع من المفاعلات للتكدية من عدم وصولنا إلى هذه الحدود بهدف التأكد من أخذ وسائل الأمان المختلفة الموجودة في المفاعل من القيام بمهامها في تأمين المفاعل من تلك الحوادث والتكد من صحة القيم المهيئة مسبقاً.
دراسة تلك النوعية من المفعالات البحثية كما تم التحقق أيضا من صلاحية النتائج بمقارنته بالنتائج المقررة والمسجلة بواسطة أجهزة المراقبة والتحكم بغرفة تشغيل المفاعل تحت الظروف المختلفة من القدرة والحالة الجوية وفي الحالة الأنتقالية وأظهرت النتائج توافقا إلى درجة كبيرة في تلك المرحلتين المستمرة والأنتقالية.

ويمتبقي هذا النموذج الرياضي على دائرة تبريد مفاعل مصر البحثي الثاني والذي يعتمد في تبريده على برج تبريد يحتوي على عدد 6 وحدات (خلايا) من النوع التبخيري المفتوح ذو التلامس المباشر بين الهواء والماء والسعة التبريدية لكل بقية 4.2 ميجاوات وعند سربان كلي بقيمة 2700 متر مكعب/ساعة حيث يتم عمل محاكاة حادثة أنهيار تدريجي لمستوى الحرارة بواسطة أخراج هذه الوحدات من المنظومة بهدف محاكاة انقطاع التيار عن المرور أو خروجه من التشغيل لسبب طارئ وهو ما قد يتعرض له المفاعل بشكل مفاجئ مما أضطرنا إلى محاكاة ونموذج هذه الحالة للتنبؤ بدرجات الحرارة التي ستأول إليها قلب المفاعل نتيجة هذا الخروج القصري والمفاجئ لمراوح التبريد من نظام التبريد مع توضيح دور نظام أمان المفاعل والتي يعول عليه حماية المفاعل سواء عن طريق فصل المفاعل فصلا طارنا أو تقليل القدرة الحرارية بمقدار 20% من القدرة النظيفة لتخفيض درجات الحرارة محل الدراسة.

ويعتمد المفاعل على مبادلات حرارية من النوع ذات الألواح والتي تعمل بنظرية السريان العكسي بين الماء الساخن القادم من قلب المفاعل والماء البارد القادم من برج التبريد والتي يتم محاكاتها أيضا في النموذج الرياضي المبكر.

يتكون قلب المفاعل من مصفوفة 5×6 بأجمالي عدد قنوات يساوي 30 قناة يتواجد بها عدد 29 قضيب من الوقود النووي ويحتوي كل قضيب على عدد 19 لوح رقيق من اليورانيوم المغلف بالألومينيوم.
تقدم هذا البحث أبتكر نموذج رياضي لمذكحة ومحاكاة نظام تبريد مفاعلات الأبحاث لتتنبأ بدرجات الحرارة المختلفة لقلب مفاعل بحثي ومكونات نظام تبريده كالمباذلات الحرارية وبرج التبريد والتي تلعب دوراً هاماً واساسياً في تحديد كفاءة نظام التبريد وأمكانية أحتوائه للحمل الحراري المتولد في قلب المفاعل. ويقوم النموذج الرياضي المبتكر بتنبؤ درجات الحرارة الصغيرة لقلب المفاعل والنتائج داخل اقطاب وقود المفاعل سواء كانت لأجزاء وقود اليوانيوم وغلاف الألومنيوم المحيط به وسائل التبريد الملاصق له في حالة تشغيل المفاعل أبتداء من المرحلة الأنتقالية وحتى يصل بهذا المتغيرات الي حالة الاستقرار وذلك من خلال الحل العددي لمعادلات الطاقة تحت الظروف المستقرة والظروف الأنتقالية. كما أن هذا النموذج الرياضي المبتكر يستخدم في محاكاة حادثة فقدان مستودع الحرارة النهائي والتداعيات الناتجة عنه من ارتفاع درجات حرارة في القنوات الساخنة لقلب المفاعل ومقارنتها بالحدود الحرارية المسموح بها في مثل هذا النوع من المفاعلات.

وقد تم تطبيق النموذج الرياضي على مفاعل مصر البحث الثاني حيث يصنف هذا المفاعل ضمن مفاعلات الأبحاث وأختبار المواد وقد تم البدا في أنشائه عام 1992 وتم الانتهاء من أنشطته وتدشينه عام 1997. وقد تم تصميم هذا المفاعل بقدرة حرارية قدرها 22 ميجاوات. ويتكون نظام التبريد لهذا المفاعل من نظامين أحدهما النظام الأولي والآخر هو النظام الثاني. ويعمل نظام التبريد بالطريقة الجبرية باستخدام طلبات طرد مركزية يتم التحكم في تشغيلها كلها أو بعضها طبقاً لقيمة القدرة الفعلية للمفاعل على اساس:

1- تشغيل المفاعل بنصف الحمل بقيمة 11 ميجاوات وهو ما يسمى ريجيم 1.
2- تشغيل المفاعل بقدرته القصوى بقيمة 22 ميجاوات وهو ما يسمى ريجيم 2.

وقد تم التحقق من صحة وصلاحية النموذج المبتكر من خلال مقارنة النتائج التي تم الحصول في المرحلة المستقرة وبرامج الباريت للتشغيل المستقر المتخصص في
ملخص الرسالة

تحليل ومحاكاة هيدروحرارية لنظم تبريد مفاعلات الأبحاث

يتناول هذا البحث إبتكار نموذج رياضي لمحاكاة فقدان مستودع الحرارة النهائي في مفاعل إختبار مواد نموذجي. ويتضمن النموذج ثلاثة نماذج فرعية متفاعلة مع بعضها لقلب المفاعل والتبادل الحراري وبرج التبريد. وقد تم إثبات صلاحية النموذج أمام الكود بارت للتشغيل المستمر والتحقق منه باستخدام بيانات تشغيل المفاعل في الحالات الإنشائية. بعد ذلك تم استخدام النموذج لمحاكاة أداء المفاعل أثناء فقدان مستودع الحرارة النهائي. وقد تم المحاكاة لنظامين تشغيل سمية: النظام (I) بقدرة 11 ميجاوات مع تشغيل ثلاث وحدات من برج التبريد، والنظام (II) بقدرة 22 ميجاوات مع تشغيل ست وحدات من برج التبريد. في النظام (I) أجريت المحاكاة في حالات تعطيل 1 و2 و3 ووحدات من برج التبريد بينما في النظام 2 فقد أجريت المحاكاة في حالات تعطيل 1 و2 و3 و4 و5 و6 ووحدات من برج التبريد. وقد أجريت المحاكاة تحت الظروف المحمية حيث قام نظام حماية المفاعل بعمل أمان يسمى إنقاذ القدرة لتقليل قدرة المفاعل بمقدار 20% عندما تصل درجة حرارة المبرد 43 درجة منوية عند مدخل قلب المفاعل كما يتم إيقاف قصرى للمفاعل عندما تصل درجة حرارة المبرد 44 درجة منوية عند مدخل قلب المفاعل. وقد تم مناقشة وتحليل النتائج التي توصل إليها النموذج المبتكر.

وقد تناولت الرسالة أيضا دراسة هيدروحرارية لحدود الأمان المتمثلة في ظواهر ثلاث تعبر عن مراحل مختلفة للغليان وهي مرحلة بداية الغليان النوى ومرحلة بداية السريان الغير مستقر ومرحلة ما بعد الغليان النوى وحساب معالات أمان المفاعل لحدث تلك المراحل. وتبين من الحسابات عدم حدوث تلك المراحل أثناء فقدان أبراج التبريد وذلك نظرا لقيام النظام المنسلول عن حماية المفاعل بأطفاء المفاعل تفاديا لذلك. كما تم حساب درجات حرارة ماء التبريد داخل قلب المفاعل وثبت بعدها عن القيم الحرج.
تحليل ومحاكاة هيدروحرارية لنظم تبريد مفاعلات الأبحاث

إعداد

م/ هشام حسنين علي الخطيب

رسالة مقدمة إلى كلية الهندسة بشبرا-جامعة بنها

كجزء من متطلبات الحصول على درجة الدكتوراة

في الهندسة الميكانيكية

يعتمد من لجنة الممتننين

1- د/ محمد فائق عبد ربه (متحنًا داخليًا)
استاذ متفرغ بقسم الهندسة الميكانيكية- كلية الهندسة بشبرا-جامعة بنها.

2- د/ محمد محمود محمد الفوال (متحنًا خارجيًا)
أستاذ الهندسة الميكانيكية- هيئة الرقابة النووية والاشعاعية.

3- د/ كرم محمود الشاذلي (متحنًا ومشرفًا)
استاذ بقسم الهندسة الميكانيكية- كلية الهندسة بشبرا-جامعة بنها.

4- د/ ماهر جميح أحمد حجازي (متحنًا ومشرفًا)
أستاذ متفرغ بقسم الهندسة الميكانيكية- كلية الهندسة بشبرا-جامعة بنها.

5- د/ صلاح الدين بلاه المرشدي (متحنًا ومشرفًا)
رئيسي قسم المفاعلات الذرية- مركز البحوث النووية- هيئة الطاقة الذرية.

كلية الهندسة بشبرا-جامعة بنها

2013
تحليل ومحاكاة هيدروحرارية لنظم تبريد مفاعلات الأبحاث

إعداد

م/ هشام حسانين علي الخطيب

رسالة مقدمة إلى كلية الهندسة بشبرا - جامعة بنها
كجزء من مطالبات الحصول على درجة الدكتوراة في
الهندسة الميكانيكية

تحت إشراف

أ. د. كرم محمود حسن الشاذلي
قسم الهندسة الميكانيكية
كلية الهندسة بشبرا - جامعتها بنها

أ. د. صلاح الدين بلال المرشدي
قسم المفاعلات الذرية
هيئة الطاقة الذرية

كلية الهندسة بشبرا - جامعة بنها
القاهرة - جمهورية مصر العربية

2013