International Journal of Chemical, Materials and Biomolecular Sciences
ISSN: 2415-6620
Vol:13, No:7, 2019

Study of Temperature Distribution in Coolant Channel of Nuclear Power with Fuel Cylinder Element Using Fluent Software

Elham Zamiri

***Abstract*—**In this research, we have focused on numeral simulation of a fuel rod in order to examine distribution of heat temperature in components of fuel rod by Fluent software by providing steady state, single phase fluid flow, frequency heat flux in a fuel rod in nuclear reactor to numeral simulation. Results of examining different layers of a fuel rod consist of fuel layer, gap, pod, and fluid cooling flow, also examining thermal properties and fluids such as heat transition rate and pressure drop. The obtained results through analytical method and results of other sources have been compared and have appropriate correspondence. Results show that using heavy water as cooling fluid along with few layers of gas and pod have the ability of reducing the temperature from above 300 ˚C to 70 ˚C. This investigation is developable for any geometry and material used in the nuclear reactor.

***Keywords*—**Nuclear fuel fission, numberal simulation, fuel rod, reactor, fluent software.

I. Introduction

IN this research, we have examined heat distribution and hydraulic parameters and also maximum temperature of fuel rod in order to improve the safety by numeral simulation of a fuel rod in single phase and steady stream. Basically, reactors are the systems in which reactions like nuclear fission occur, huge amount of energy is produced, and this energy is converted to electricity. The sun is a natural nuclear reactor in which nuclear fusion occurs and huge amount of energy is released. In overall, fuel is in the center of reactor which released a lot of heat. At the center part of each nuclear reactor, there is a container where cleavable material (nuclear fuel) is put. The type of cleavable material in the container considering the type of reactor varies, but mainly natural 0.7% uranium is with 235 isotope and enriched 3.5% uranium with isotope 235. The method of cooling is of high importance, because there is possibility of a gap and leak of radioactivity rays, and this issue has been researched through analytical and numeral approaches [1]. One of the most important reasons of using nuclear energy is releasing a huge amount of energy with the least material. Comparing fuel efficiency, if an uranium atom is split, about 200 MeV energy is released considering the energy produced by 1 gram uranium equals to energy produced by burning 2 tons of charcoal, so we cannot ignore such huge amount of energy [2], [3]. Generally, reactors are different and each has its characteristics, but the main equipment of a nuclear reactor is mainly divided into three categories [4]-[7]:

1) A device to regulate chain reaction, such as control rods which are responsible for neutron absorption.
2) Moderator which surrounds the fissionable material inside center container. This reactor enables the possibility of their interaction with the core and triggers the fission process and thus the continuation of chained interaction.
3) A device for heat transfer as a result of fission process and chained reaction outside of interior part of reactor which is used for rotation of electricity generator turbines. In this thesis, we have studies this matter.

In this study, we have discussed precise process of cooling in a water nuclear reactor under pressure. At first, we take a closer look to water reactor under pressure from importance of cooling view. The heat generated by fuel in the central core of reactor is absorbed through main fluid which is usually water. This boiling water is termed hot base. This hot based is directed toward turbine to convert thermal energy to electricity with high efficiency. Then, with drop of fluid temperature which is called cold base and after passing condenser is returned to the cycle.

One of the most important processes in cooling cycle is thermal convertor. This equipment ensures proper heat transfer. This equipment can do the heat conversion action with two parallel fluids or vice versa in metal tubes. Different types of this equipment which are called thermal, plate, shell and bladed tube heat convertor are available. Usually, in heavy water nuclear reactors, tube – shell heat convertor is used where fluid flow passes through main channel and along the flow of fuel rods in the shell conducts the transfer heat action. The simulation action contributes great help to the design of this equipment, as efficient flow rate, flow type, channel size and heat transfer will be found. Pekdemir et al. [8] measured passing flow rate from different cross sections and speed distribution rate drop pressure to reach efficient cooling. They used the experimental method of particles pursuit. Their results are presented as experimental applied coefficients which are used for reactors with different sizes. In another research [9], they investigated the measurement in reactor with tube thermal convertor element. They showed that one of the most important effects in cooling is the drop pressure rate in thermal convertor, and precise measurement of this parameter is necessary. Wang et al. [10] also did the experimental investigation of heat transfer in a reactor with tube – shell heat

Elham Zamiri is M. A. in Nuclear Engineering from Azad Islamic University of Arsanjan Branch, Iran, Islamic Republic Of (e-mail: e.zamiri9090@gmail.com).

convertor ,they suggested the use of channel for the gaps between convertor plates which can improve efficiency rate cooling up to 25%. Researches on analytical methods also have been presented, as an example, Kapale and Chand [11] presented an analytic model. They predicted precise drop pressure for reactor equipped with tube – shell convertor element. Also Vera-Garcia et al. [12] developed these results for different geometry. Since experimental research was expensive and analytic researches were limited to specific geometry, numerical simulation researches without building cost or geometry limitation were conducted by researchers. For example, Ozden and Tari [13] used fluids dynamic methods for a small thermal convertor of nuclear reactor and obtained the whole pressure and temperature distribution and then performed the accreditation. Sarchami [14] investigated a heavy water nuclear reactor using Fluent software and showed the whole drop pressure and temperature distribution for CANDU nuclear reactor along with accreditation in three dimensions with the help of parallel process. In this regard, Aminifar and Mohammadi Pourfar [1] analyzed the heat in fuel rod of a heavy water nuclear reactor and showed that simulation of a reactor greatly helps to proper design of a reactor. They used Fluent software to simulate a fuel rod with pod and accredited his results with Cobra 3 software. Amir et al. [15] studied and analyzed some thermohydraulic parameters in cooling reactor channels *IPR-RI* including characteristics of radial and axial temperature, temperature speed, mass flow rate, mass and number flux. Some of the results were compared with the theoretical predictions, and it was expected that variables follow the power distribution in the heart of reactor. Sinha and Dulera [16] investigated the performance of nuclear reactor which their initial materials are from carbon in Bhabha atomic research centre (BARC) under high temperature. Amidst their researches, they found out that dimensional radiation step to step and reduction of heat and mechanical properties are important factors. Results including the design of these high temperature reactors have been in a general state.

Adjei et al. [17] investigated equality of transient and unstable graphs in fuel rod in a MNSR reactor while losing the whole cooling. This transient action was investigated analytically using Bessel functions and also expansion of transient temperature equations law for a fuel element. The obtained results during sudden loss of the whole cooling showed that a transient high temperature distribution in center of fuel element oppose to fuel pod level which recorded the least temperature reduced to distribution of transient temperature from center of fuel element until fuel pod level and followed from a descending graph parabolic which increased with time. During a sudden blackout until there is no generated or lost heat ,the rate of cooling of the fuel element is directly proportionate with time.

II. Equations Present in the Problem

General equations present in the problem include continuity equations, momentum and energy. The main principle which is used in fluids mechanic is the principle of mass survival. This principle suggests that mass is neither produced nor destroyed but is showed for a general state using continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho v) = 0 \tag{1}$$

Characteristics and features of a fluid in fluids mechanic is not only determined by having continuity equation but survival principle, momentum, or second law of Newton should be shown about it. Momentum is the multiplied of mass in speed. Second law of Newton expresses that result of forces which affect an object equals pure momentum changes.

Considering incompressible flow and stable viscosity supposition, general form of equation of momentum survival is as follows. This equation is known as Navier Estokes equation.

$$\rho \frac{DV}{Dt} = \rho f - \nabla P + \mu \nabla^2 V \tag{2}$$

where V speed vector, P represents pressure, f volume forces and μ is for viscosity. D/Dt represents material derivative and is defined as:

$$\left(\frac{D\varphi}{Dt} = \frac{\partial \varphi}{\partial \mathrm{x}} + \vec{\mathrm{V}}, \nabla \varphi\right) \tag{3}$$

Considering that the flow under investigation in this process is turbulence flow, it is required that we investigate equations in a turbulence format. In order to obtain equations present in the flow in a turbulence state, first we rewrite equations for moment quantities, which means average quantities and frequency quantities. Then, from two sides of the equation, we do moment average. In this state, we should consider point that if we have equality for moment equations, this equality is also applied for moment average (for a specific period of time). In the end, we simplify the equations to the point where average moment quantities emerge. Continuity equation for compressible fluid in turbulence flow is expressed as follows:

$$\frac{\partial}{\partial x_i}(\overline{\rho\, u_\iota}) + \frac{\partial}{\partial x_i}(\overline{\rho'\, u_\iota'}) = 0 \tag{4}$$

Momentum equations for turbulence flow are expressed as:

$$\rho \left[\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j}\right] = \bar{B}_\iota - \frac{\partial \bar{p}}{\partial x_i} + \quad \frac{\partial}{\partial x_j}\left[\mu \frac{\partial \bar{u}_i}{\partial t} - \rho \overline{u_i' u_j'}\right] \tag{5}$$

The only difference of momentum equation above with momentum equation with moment quantities is the addition of the last phrase in the right side of the equation. This phrase is termed turbulence stress or Reynolds stress. The only difference of slow flow equations with the turbulence ones is also this very phrase. Generally, physically speaking this phrase is not a stress but expresses the effect of inertia exchange (momentum). The problem includes a fuel rod which from exterior layer of rod connects with fluid, middle layer of pod, gap layer and fuel core. The equation of overall heat transfer equates the rate of heat transfer directed also with radiance heat transfer axis but because we consider radiance in very high temperature (plasma flow) is ignored and only focused on directed heat transfer [1]. Considering directed heat transfer for energy equation:

$$\mathrm{Q_{total}} = \mathrm{Q_{cond}} + \mathrm{Q_{rad}} \tag{6}$$

$$\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial T}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 T}{\partial \emptyset^2} + \frac{\partial^2 T}{\partial z^2} + \frac{q'''(z)}{K} = \frac{1}{\alpha}\frac{\partial T}{\partial t}$$

III. Process of Simulation and Algorithm for Resolving Problem

In this research, Fluent software of dynamic simulation of fluid calculation is used. Resolving equations in Fluent is done with both separation solution and continuity solution. Fluent software can use either of the methods above to solve any integral equations present in the problem for survival (mass and momentum), and if necessary, for energy and other scalars such as turbulence. In both cases, a technique based on control volume is used:

Dividing the area of calculating operation into separate control volumes by using an integral computation network including dominating equations for the problem in control volumes for defining algebraic equations relating to unknowns such as melocities pressure and temperature, and making separated equations linear and solving resulted linear equations systems for reaching corrected values for dependent variables. These two numerical methods are using one separating method (limited volume) but linearizing method and solving separated equations are different.

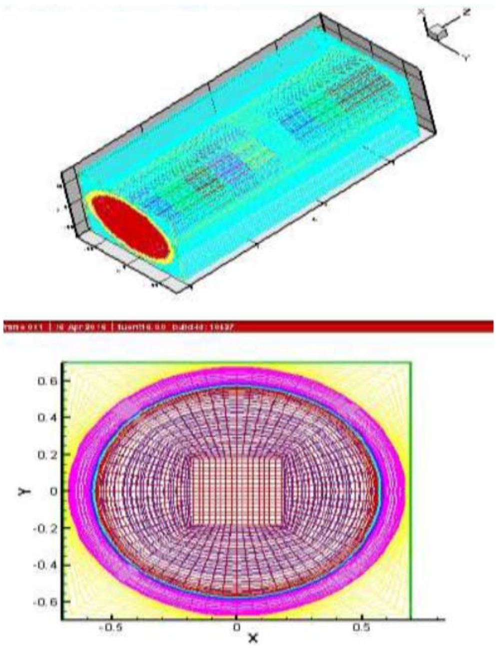

Fig. 1 Geometry of fuel rod is demonstrated

A. Networking Fuel Rod and Different Layers

In this problem because of different layers meshing in few layers is required which include first layer, fuel rod layer, pod layer, gap layer, and fluid layer. In Gambit software, first different layers and fuel rod are drawn based on geometry data, and then, one level is defined for each part. After that, for each level, some nodes are defined and meshed based on formed level and node. The mesh used in this thesis, is structured rectangular. In this research, 65 layers are defined for describing different level of this geometry.

One of important issues in meshing is to evaluate quality of mesh. Gambit software has analysis quality and can evaluate quality of the mesh. The result of this meshing is presented in Table I from Gambit software for the fuel problem mesh. Based on this analysis for 42880 element more than 60% of elements have very high quality (0.1) 12 % high quality (0.2) and the rest good quality (0.3 – 0.5) and about 2% average quality. Important point in the production of mesh process is considerable, linear element in publication is not considered, and with this technique error is reduced in order to adjust parameters for resolving the problem in Fluent because we deal with heat transfer the energy model must be active. Also turbulence model *k-ε* is examined for simulation.

TABLE I
RESULTS FROM GAMBIT FOR EXAMINING THE QUALITY OF MESH

From value	To value	Count in range	% of total count (428800)
0	0.1	261248	60.93
0.1	0.2	54480	12.71
0.2	0.3	27664	6.45
0.3	0.4	38272	8.93
0.4	0.5	35136	8.19
0.5	0.6	12000	2.80
0.6	0.7	0	0.00
0.7	0.8	0	0.00
0.8	0.9	0	0.00
0.9	1	0	0.00
0	1	428800	100.00

IV. Discussion and Results

In this section, we examine the obtained results from Fluent software and compare with analytical results. First supposition for heat distribution in fuel rod is the cosine flux distribution. Such cosine flux distribution, also affects heat distribution and such heat distribution should be resulted. Two other suppositions are considered in this simulation, first, cross semester is removed compared with linear term, and from radiance distribution because of low temperature, and about first supposition sentence along *Z* axial equals $(\frac{\partial^2 T}{\partial Z^2})$ and along radial equals $\frac{1}{\mathrm{r}}\frac{\partial}{\partial r}(\frac{\partial T}{\partial r})$. With defining two terms based on numerical method, it is resulted that the limited difference graph is a true supposition. Since radiation factors in very high temperatures become activated, they do not affect in calculation of the current problem. Number of nodes in longitudinal direction, boundary conditions of cosine heat flux in fuel is considered to mass fluid flux according to Table II for a rod.

TABLE II
HEAT FLUX BOUNDARY CONDITIONS AND NUMBER OF NODES FOR A ROD

Cosine heat flux in fuel to number of modes in mass fluid flux direction	
z/L	$\frac{q''}{q''_{ave}}$
0	0.273
0.1	0.636
0.2	0.951
0.3	1.195
0.4	1.349
0.5	1.401
0.6	1.349
0.7	1.195
0.8	.951
0.9	.636
1	.273

The results of simulation are examined for two states of unreal stable flux and real variable flux. In the first state, it is supposed that heat transfer flux (maximum volume) is stable. In Fig. 2, temperature counter in a fuel rod level is drawn considering different layers and stable flux. In Fig. 3, counter of speed changes in cross section of a fuel rod is demonstrated. As one can see that maximum temperature in center of a fuel rod is more than an external layer in contact with fluid and in all sections we have a constant temperature distribution because stable distribution in the whole rod is present. There is no such distribution in a fuel rod real reactor. In real state of heat or speed, distribution in a fuel rod is different in each section.

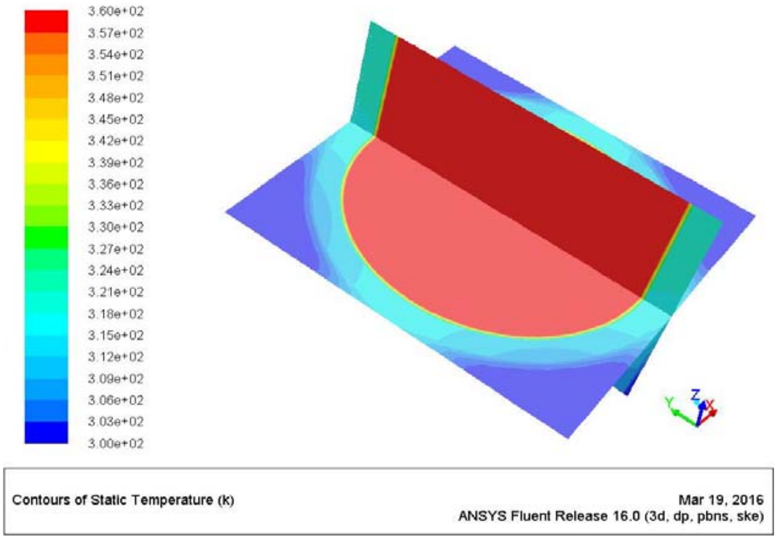

Fig. 2 Fluid tempreture counter with supposition of stable flux

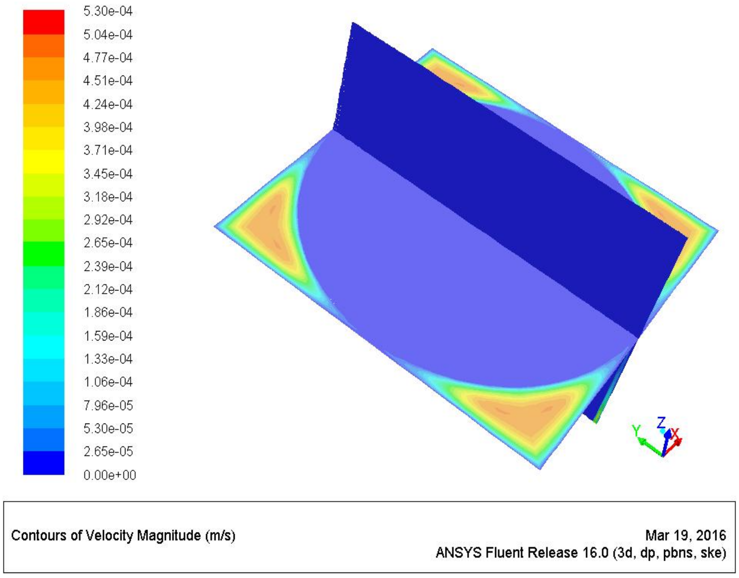

Fig. 3 Speed counter in fuel rod with supposition of stable flux

International Journal of Chemical, Materials and Biomolecular Sciences
ISSN: 2415-6620
Vol:13, No:7, 2019

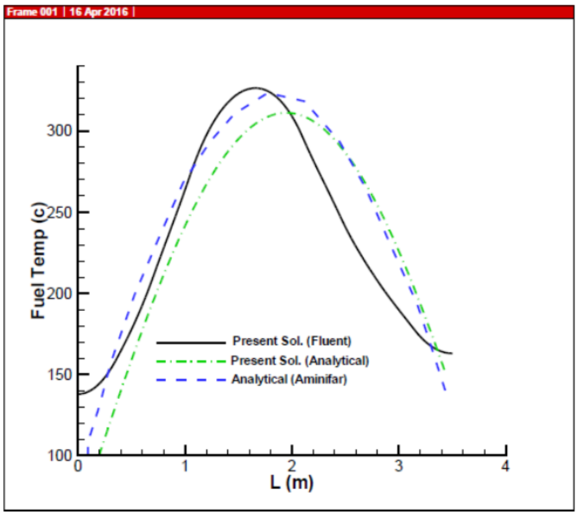

Fig. 4 Fuel rod temperature changes based on rod longitude graph

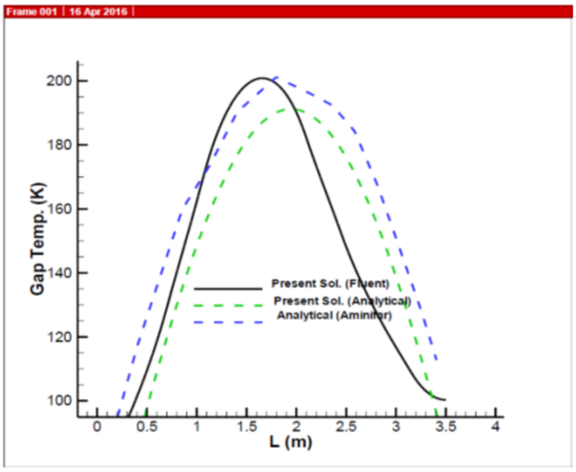

Fig. 5 Pod temperature changes based on rod longitude graph

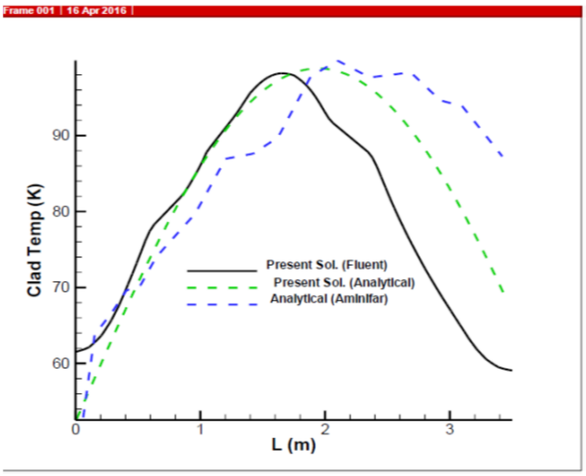

Fig. 6 Gap temperature changes based on rod longitude graph

As mentioned, heat distribution in longitudinal direction of fuel rod is totally variable and usually has triangular distribution. In Fig. 4, temperature changes of fuel rod are presented with analytical and reference analytical results [1]. This examines for pod temperature distribution in Fig. 5, gap temperature distribution in Fig. 6, and fluid temperature distribution in Fig. 7 is presented. Based on the obtained results, we can point out to reduction of temperature in each stage to the least degree from about 300◦ to about 70◦. In other words, if the proper cooling is done, proper reduction is achieved and the potential dangers from leak and radiation and radioactive material is reduced.

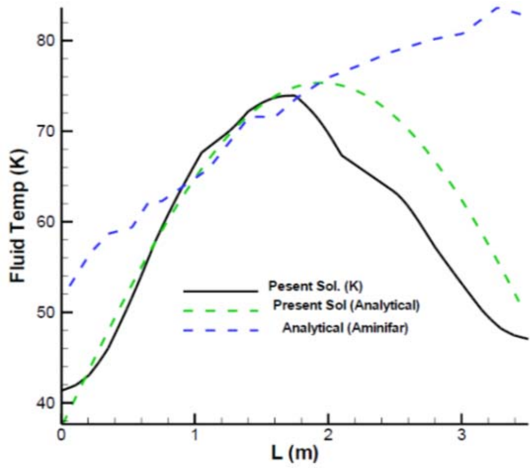

Fig. 7 Fluid temperature changes based on rod longitude graph

V. Conclusion and Suggestions

In this research, we discussed simulation of cooling problem of a fuel rod. Cooling problem of fuel rod is of important topics in a nuclear reactor because if the cooling is not done properly there is possibility of fission and leakage of radioactive material. Furthermore, in second chapter we discussed cooling with help of fluid and in this regard the method of analytical investigation was presented. The issue of fuel rod was discussed in four layers of fuel rod, pod, pod, gap and fluid. At first, the analytical results were developed and compared with available results for different layers which has a good correspodence. Moreover, considering conditions of the issue, for a fuel rod in different layers, simulation is presented in two states of stable flux (unreal state) and variable flux (real state). It was demonstrated that in the state of variable flux in different cross sections parameters for fluids such as temperature is different. The obtained results along fuel rod was accredited with analytical and related reference results. Results showed that with cooling fluids it is possible to reduce the maximum temperature from 300◦ to 70◦.

References

[1] H. Aminifar, M. Mohammadi Pourfar, "heat transfer analysis in fuel rod of nuclear power plant.", University of Tabriz Magazine, Mechanic engineering, Volume 32, Number 2, 1384.
[2] A. Fentiman, "Nuclear Power Past, Present, and Future," Purdue University, 2012.
[3] The history of nuclear energy, DOE, NE-008.
[4] Overview of Nuclear Reactor Systems and Fundamentals.

[5] U.S. Energy Information Administration (EIA). New Reactor Design.
[6] S. Place "Nuclear Reactor Types" An Environment & Energy FactFile provided by the IEE, 2005.
[7] Types of Nuclear Reactors, Institute for Energy and Environmental Research Celebrating 25 years of science for democracy.
[8] T. Pekdemir, T. W. Davies, L. E. Haseler, A. D. Diaper, "Flow distribution on the shell side of a cylindrical shell and tube heat exchanger", Int. J. Heat and Fluid Flow, 14(1),1993.
[9] T. Pekdemir, T. W. Davies, L. E. Haseler, A. D. Diaper, "Pressure Drop Measurements on the Shell Side of a Cylindrical Shell-and-Tube Heat Exchanger", J. of heat transfer engineering,15(3).1994.
[10] S. Wang, J. Wen, Y. Li, "An experimental investigation of heat Transfer enhancement for a shell-and-tube heat exchanger", J. of Applied Thermal Engineering,29,2433-2438, 2009.
[11] U. C. Kapale, S. Chand, "Modeling for shell-side pressure drop for liquid flow in shell-and-tube heat exchanger", International Journal of Heat and Mass Transfer,49:601-610, 2006.
[12] F. Vera-Garcia, J. R. Garcia-Cascales, J. Gonzalvez-Macia, R. Cabello, R. Llopis, D. Sanchez, E. Torrella, "A simplified model for shell-and-tubes heat exchangers: Practical application", J. of Applied Thermal Engineering, 30, 1231-1241,2010.
[13] E. Ozden, I. Tari, "Shell side CFD analysis of a small shelland- tube heat exchanger, J. of Energ y Conversion and Management ,51,1004-1014,2010.
[14] Araz Sarchami, Investigation of Thermal Hydraulics of a Nuclear *Reactor Moderator, PhD thesis,* Mechanical and Industrial Engineering Department University of Toronto, 2011.
[15] Amir Z. Mesquita1, Daniel A. P. Palma2, Antonella L. Costa3, Claubia, Pereira3 Maria A,.F Veloso3.F. . and Patrícia A L. Reis3, Experimental, Distribution of Coolant of in the IPR-R1 Triga NuclearReacto r Core, International Nuclear Atlantic Conference- INAC2011, Belo Horizonte, MG , Brazil, October24-28, 2011.
[16] R. Shinha, I. Dulera, Carbon based materials- application in high temperature nuclear reactors ,Inidian Journal of engineering & materials engineering & Sciences , Vol.17 (2010), pp.321-326.
[17] C. A. Adjei, A. G. Ayensu, E. H. K. Akaho, F. K. Quashie, A. B. Asumadu-Sakyi and E. O. "Amartey, Equalisation of Transient Temperature Profile Within the Fuel Pin of a Miniature. Neutron Source Reactor (MNSR) During Total Loss of Coolant," Research Journal of Applied Sciences, Engineering and Technology, vol. 2(2010), PP 661-668.