Original Research Article

Validation of ANSYS model of experimental test rig simulating the Flow Inversion in RRs

Abstract

An experimental test rig was built to simulate the flow inversion in natural circulation loop in Research Reactores (RRs) in an attempt to understand the built up of natural circulation in RRs pool type upward flow after pump coast down due to power loss by **Abdel-Latif et al. [1].** The test rig is made up of two vertically oriented branches, one of which contains two rectangular, electrically heated, parallel channels that simulate the core. The other branch represents the portion of the return pipe that is involved in the development of core natural circulation.many experimental runs at different conditions as branch's intial temperatureis performed.the channel's coolant and surface temperatures were performed.In this study, the thermal hydraulic behaviour of the test rig is complemented by theoretical analysis using ANSYS Fluent 17.2 code. the ANSYS Fluent model is validated against the measured values. The test rig is typically nodlized and an input dick is prepared. In spite of the low pressure of the test rig, the results show that ANSYS Fluent 17.2 qualitatively predicts the thermal hydraulic behaviour and associated flow inversion phenomenon of such facilities.Quantitatively there is a difference between the perdicted and measured values especially the channel's surface temperature.

KEYWORDS: *Research Reactores, Parallel Channel, ANSYS FLUENT, Natural Circulation, Flow Inversion*

1. Introduction:

The design objectives of research reactors include adequate margins against critical phenomena such as critical heat flux, flow instability, and flow inversion.Takeda, et al. [2] in 1987 have developed experimentally and analytically natural circulation in parallel channels with different heat input. A flow rate and direction have been obtained in each channel depends on the time history of the channel temperature. It was found that the flow rate and direction in many parallel channels are affected by the addition of weak forced circulation. Analytical model has been predicted for the natural circulation of the flow including the reversing. Chang and Stan [3] in 1993 have carried out experimentally steady state CHF for conditions simulating natural convection conditions in a vertical rectangular flow channel with L/D ratio of 154 both upward and downward flow. **Ouyang andYang** [4] in 1994 investigated the capability of natural circulation cooling of reactors following loss of cooling accident using a test facility. A vertical heated tube and an unheated by pass channel were used with initial up flow and down flow conditions. Both experimental results and the RELAP5 have compared. A transient phenomenon, from forced convection through flow reversal into natural circulation cooling, has observed.The analytical results have showed that a natural circulation cooling pattern can be established and that a significant cooling period can be maintained for both the initial up flow and the initial down flow conditions. The RELAP5 simulation program predicts the transient phenomena from forced convection into natural circulation cooling very well. Both the initial flow direction and the coast down rate have effects on the development. The two-phase natural circulation cooling limit is higher for the test started with up flow forced convection than for the tests with initial down flow. In addition, the Natural cooling circuit (NCC) developed from an initial down flow system subject to greater flow oscillation than that developed from an initial up flow system. The pump coast down is identified as a significant factor in the establishing of natural circulation from an initially forced flow condition. The effect of pump coast down is also dependent on the initial flow direction. In case of vertical down flow, the results from the RELAP5 simulation clearly demonstrated a strong effect of the speed of flow coast down on the establishing of natural circulation cooling. The simulation results showed that for an initial vertical down flow, the faster the flow coast down, the easier for the developing of natural circulation cooling. For the test with initial up flow force convection, the effect of pump coast down rate is not significant. **Abe et al.** [5] in 1994 have studied experimentally and analytically the fundamental two-phase flow behavior under natural circulation conditions the focusing the flow patterns has been investigated experimentally, flow rate and two-phase behavior inside the vessel. The natural circulation characteristics have performed analytically. Both experimental and analytical results confirmed that sufficient and stable flow can be achieved in a BWR natural circulation system. Khattab and Mina [6] in 1999 have analyzed research reactor core thermal-hydraulics using rod and plate types fuel elements without altering the core bundles square grid spacer (68mm, side). Reactor power could be upgraded from 2 to10 MW without significantly altering the steady state, thermal-hydraulic safety margins, fuel, clad and coolant transient temperatures are determined inside the core hot channel during flow coast down using PARET code. Housiadas[7] in 2000 has simulated numerically the course of loss-of-flow transients in pool-type research reactors, with scram disabled. The analysis with a customized version has performed with the code PARET. Two-phase flow stability boundaries have been determined as function of initial reactor conditions. The author recognized that the flow instability is the basic mechanism responsible for core damage in such type of transients. A useful chart, which describes the stability region in terms of initial reactor power, initial pool temperature, peaking factor, and flow-decay time constant, has been provided. Wang and Vafai [8] 2000 have investigated experimentally and analytical the thermal performance of a flat-plate pipe during startup and shutdown operations. An analytical model developed has been used in a previous study. They have presented the effect of input power and coolingon the thermal performance of the heatpipe.

The main objectives of the present work is to investigate experimentally the thermal hydraulic transient behavior of the MTR pool type upward flow at the later stage of pump coast down which follow the reactor scram due to the loss of power .The study aims to clarify the effect of the reactor pool temperature.

2. Brief description of the experimental work:

the flow inversion has been studied experimentally with a test rig as shown in figure1was designed in reference [1] is selected as the research object, and it'sgeometric. This test rig consists of a vertically oriented test section (1), contains two electrically heated channels extended between upper and lower plenums (2&3). The upper plenum,(3) is connected to a cold water tank (4) through a vertical pipe (15). The water temperature in tank (4) is changed through a cooling circuit (8). The lower plenum,(2) of the test section is connected to hot water tank (5) through a vertical pipe (9) simulating the core return pipe. An electrical heater (6) is inserted in the tank (5) to change its temperaure. To establish a closed cooling loop the two tanks are connected through a ball valve (7). Many copper-constantan thermocouples are calibrated and axially disturbuted on the coolant channles and their heating walls.the bulid up of natural convection in the above test rig is assumed to be governed by three key parameters ; the channel power the initial left and right column temperatures. Therefore, three growps of experimental runs are performed, in each growp only one of the key parameters is changed. One of these experimental groups is used in the present study for ANSYS Fluent 17.2 code validation.this group consisting of fourruns at left column temperature of 283, 293, 303, and 313K. the other key parameters are constant at; channels power 1000 and 500Watts and the right column temperature 323K.

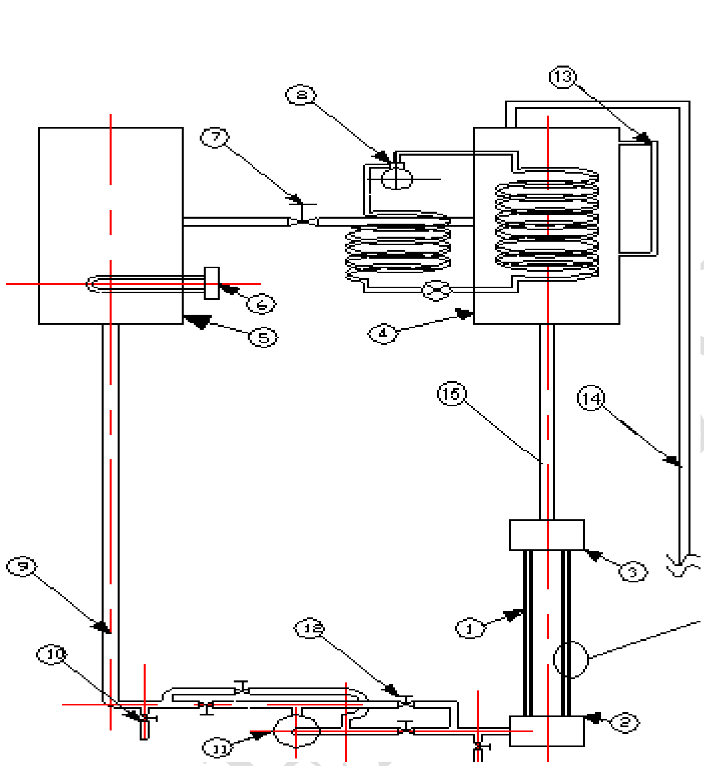


Fig. (1) Schematic diagram of the test rig [1].

1- Test section	9- Hot vertical pipe
2- Lower plenum	10- Drain
3- Upper plenum	11- Pump
4- Cold tank (pool)	12- Valve
5- Hot tank (inlet channel)	13- Water indicator
6- Electric heater	14- Water supply
7- Valve	15- Cold vertical pipe
8- Cooling circuit	

3. ANSYS Fluent 17.2 model

3.1- Description of ANSYS Fluent 17.2 computational model:

In order to model the natural circulation, the governing equations were solved using a finite volume method. The ANSYS Fluent 17.2 code [9] is used for numerical simulations. The numerical simulation exactly mirrors the experiment. The experiment is modelled with 3D-structural elements.The model consists of two vertically oriented branches, one of which contains two rectangular, electrically heated, parallel channels that simulate the core. The other branch represents the portion of the return pipe that is involved in the development of core natural circulationas shown in figure (2).

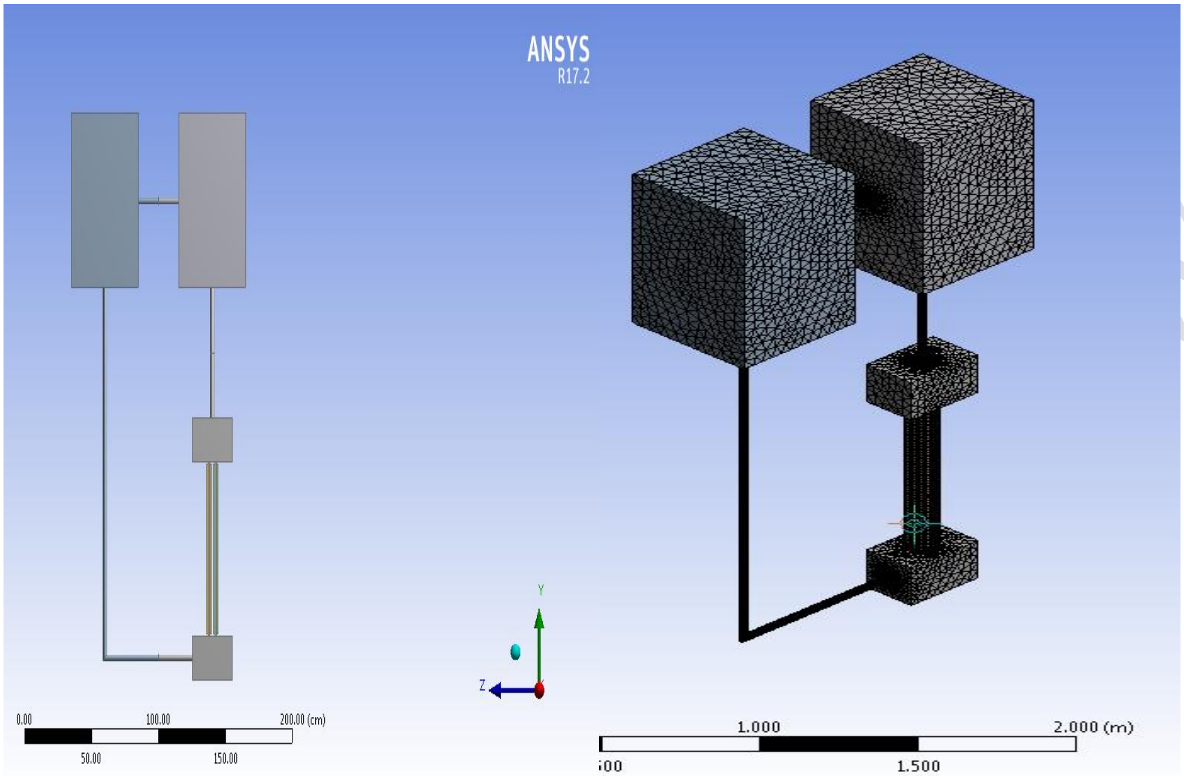


Fig. (2).Test RigANSYS ModelFig. (3). Mesh of ANSYS Model.

The theoretical predictions are validated with the experimental measurements. The mesh resolution as shown in table 1(Mesh Report),this is in line with ANSYS best practice guidelines for modeling natural convection flows as shown in Fig. (3).

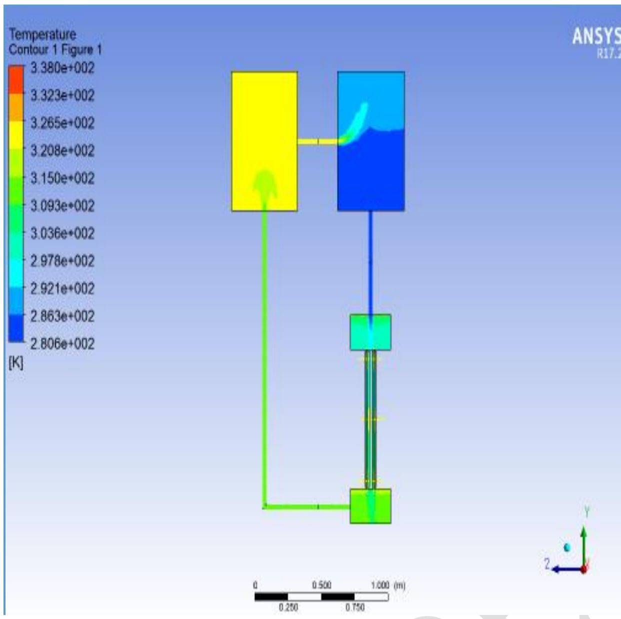
Table 1. Mesh Report

Domain	Nodes	Elements
cold leg	11486	55789
heater1 2	2612	1146
heater3 4	2642	1161
hot leg	31508	146623
main	14271	62142
All Domains	**62519**	**266861**

4. Results and discussions:

the results presented herconsisting of twosections .

4.1The first section displays the graph of the results obtained from ANSYS model. Figure (4) represents the field of temperature at the start of the developed model to show the initial conditions. Figure (5) shows the variation of the temperature through the two channels, the flow outlet from the top of the high power channel is remarked, where the flow outlet from low power channel at the bottom



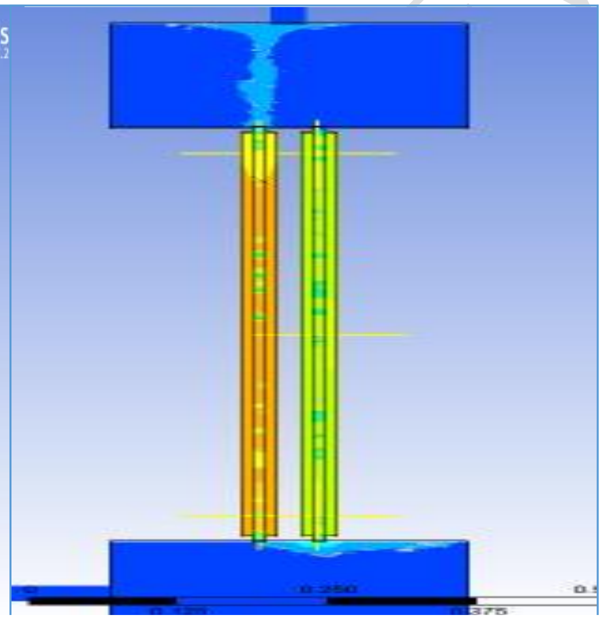


Figure (4)Temperature fields, of ANSYS Test RigModelFigure (5) contour of the temperature test section

Figure (6) representsthe variation of velocity stream Lines in the two channels. From ths disturbution of the stream lines ; out from high power channel (left side) and inlet to low power channel (right side) . The flow revesl in low power channel is conducted.

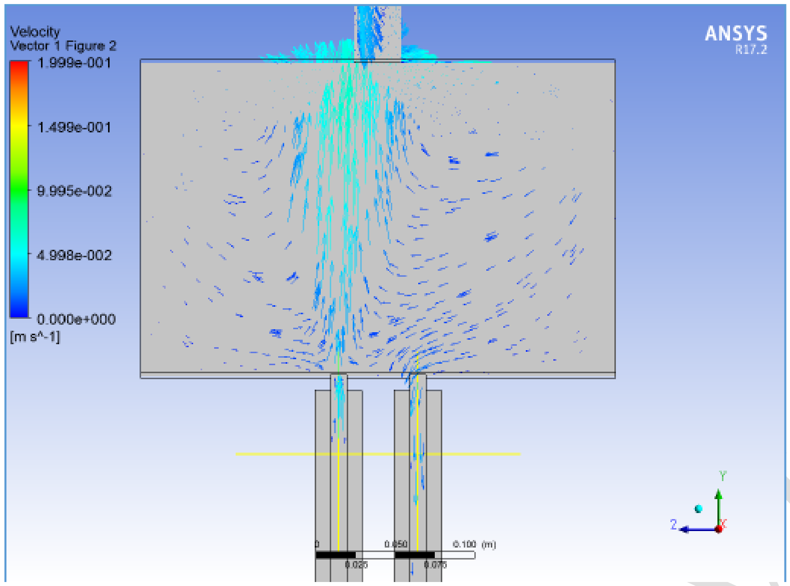


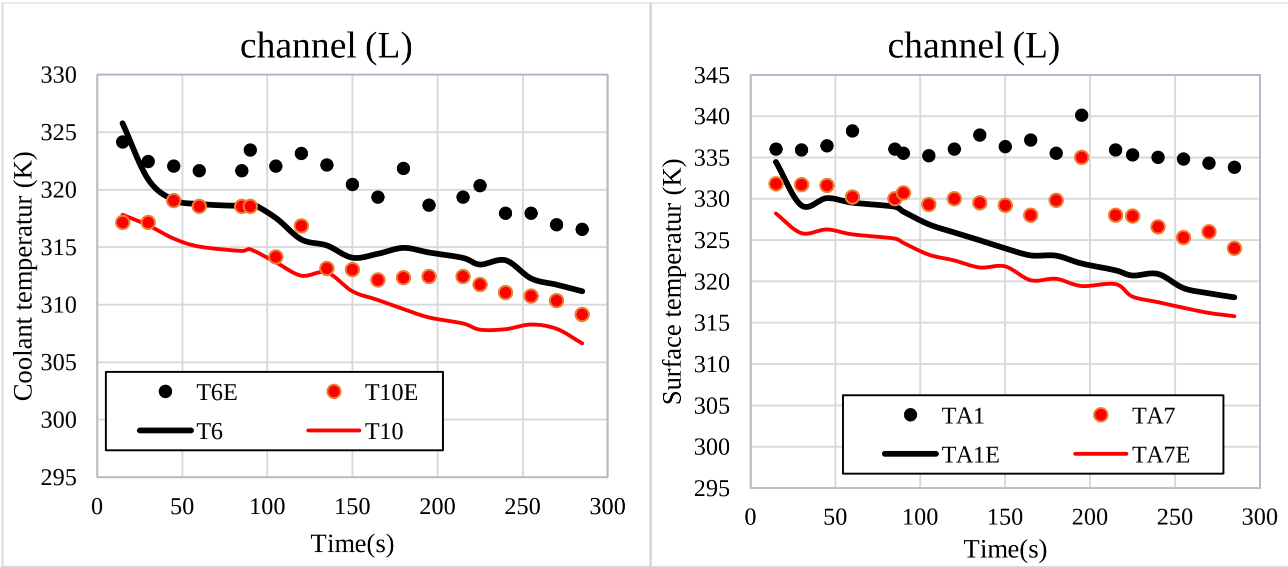
Figure (6) Instantaneous velocity stream Lines

4.2. Validation of the ANSYS FLUENT 17.2 model

Validation analysis has been performed over the studied range of pool temperatures (283-313K) in order to confirm the accuracy of the model. To do this, the data (coolant temperature) calculated by ANSYS model were compared to the measured data. Figure 8 depicts the compared inlet and outlet channel coolant temperatures at a constant power ratio =1/2. The comparisons show the good agreement between the code outputs and experimental data.

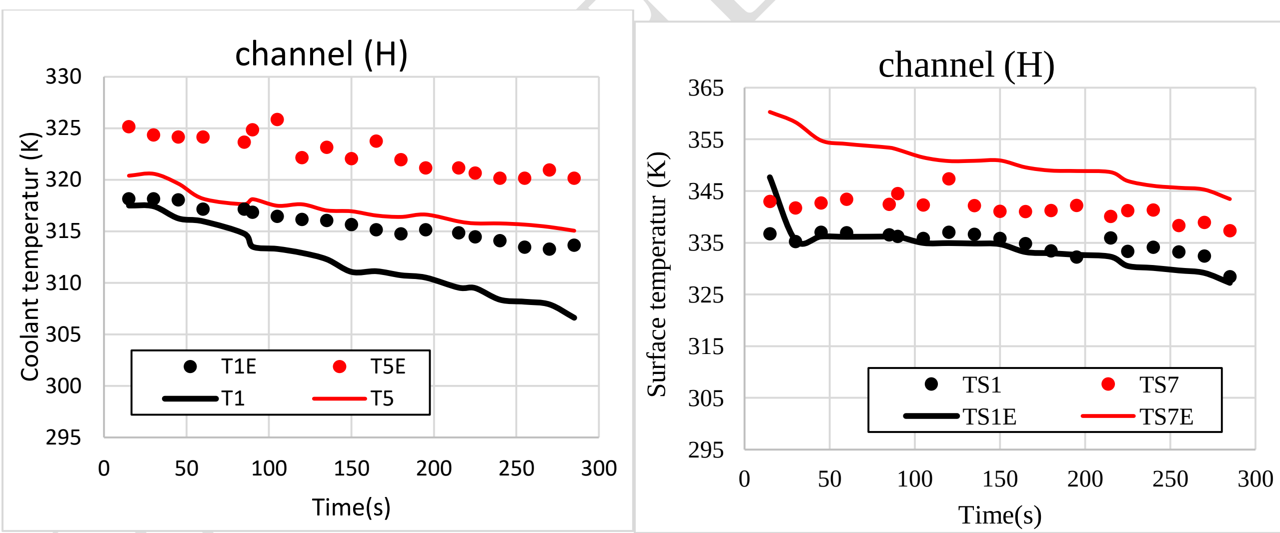
4.2.1. Comparison between experimental measurments and calculated (ANSYS Model results)

Figures (7) and (8) , show the channels coolant and surface temperatures at their lower and upper ends for right column temperatures (Tr) of 283 and 293 respectively. Generally there is a difference in the temperature values perdicted by ANSYS code continuous lines where , (T6&T10), the coolant inlet and outlet temperatures for low power channel respectively. TA1&TA7, the surface inlet and outlet temperatures respectively for the same channel. the experimental measurments represented by dots shape and denoted by(T6E&T10E) and TA1E,TA7E),



(a) (b)

In high power channel, the coolant measurmenttemperatures denoted by T1E,T5E andthe surface inlet and outlet temperatures respectivelyTS1E,TS7E. In ANSYS code continuous lines where In high power channel, the coolant temperatures T1,T5 andthe surface inlet and outlet temperatures respectivelyTS1,TS7 as shown in the next fighers



(c) (d)

Figure (7) Comparison between experimental measurements and ANSYS resultsof coolant and surface temperatures at inlet and outlet of high and low power channels for right column temperatureT_r=283K

On the other hand, from the coolant temperature , it is obvious that there is an agreement between ANSYS results and experimental measurments regarding the direction of the flow and downward flow in the low power channel (L) and upward flow in the high power channel (H), where the coolant temperature at the lower end is higher than that at upper end of (L) and vice versa in (H). Also the deivation between ANSYS results and experimental measurments decreased with increasing the right column temperature Tr as shown in figure (8).

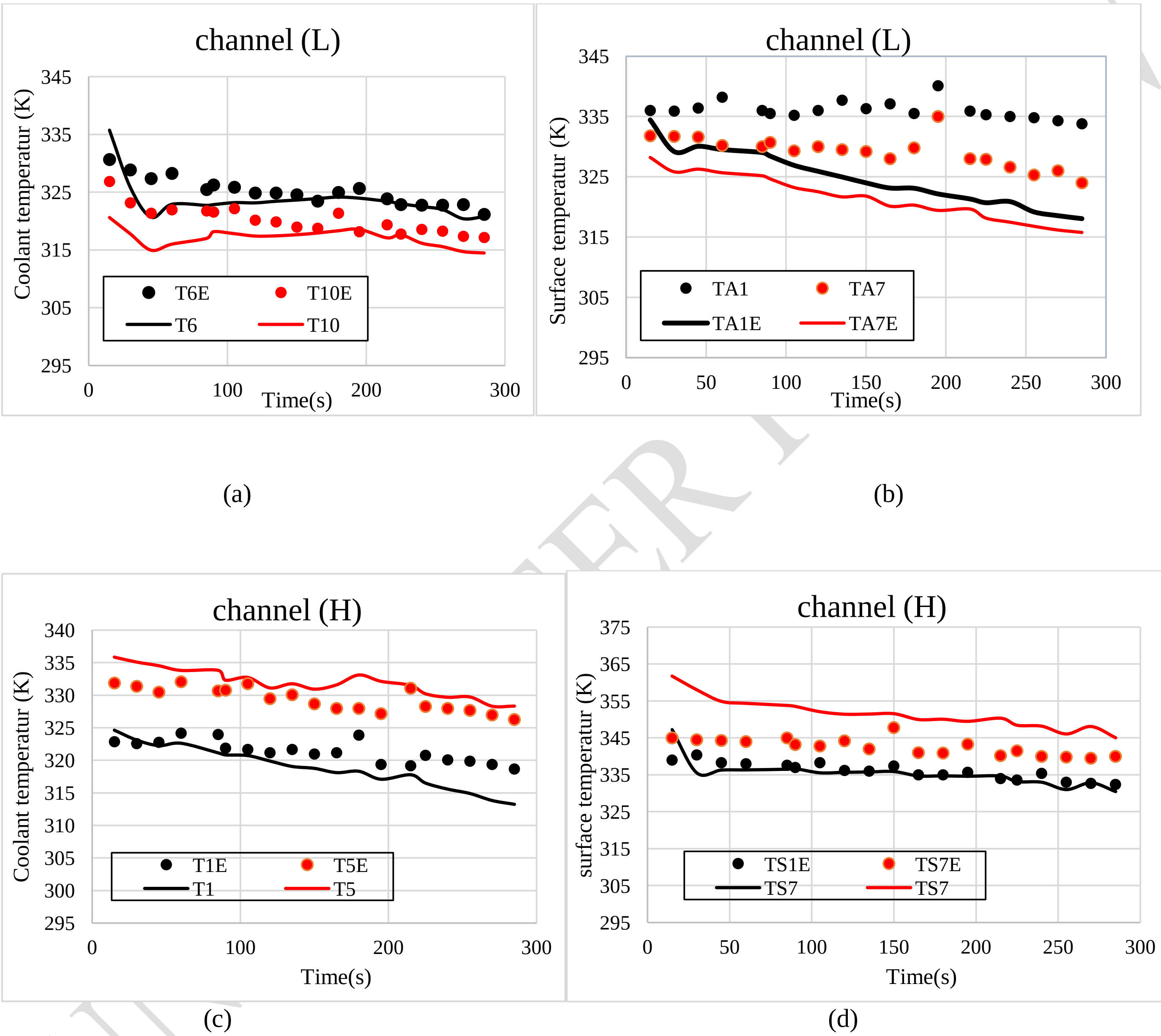
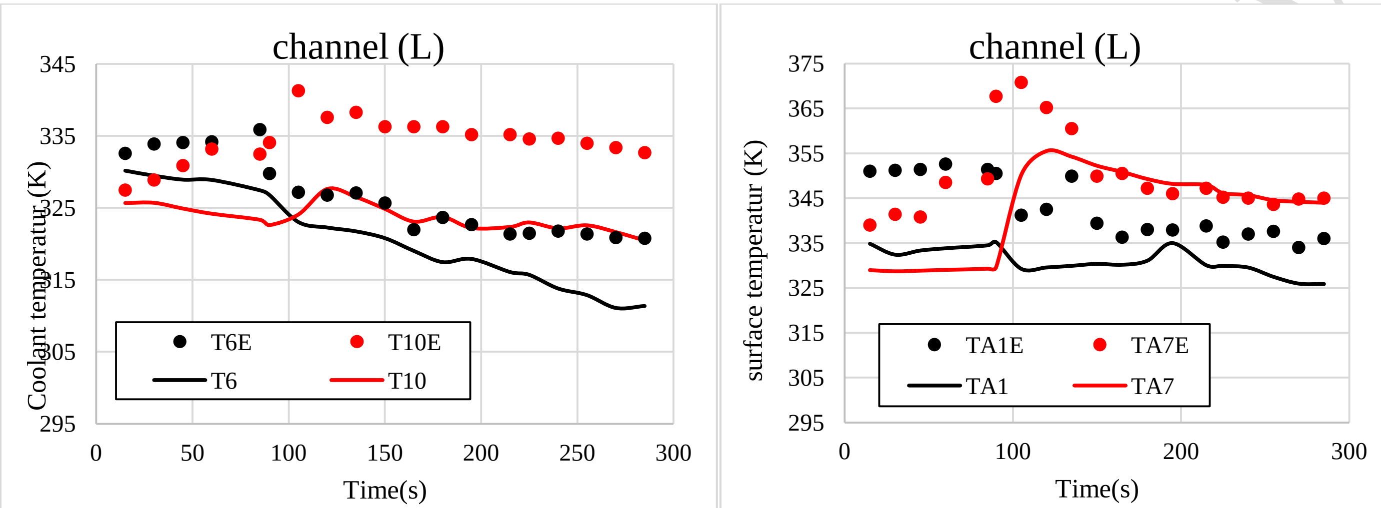


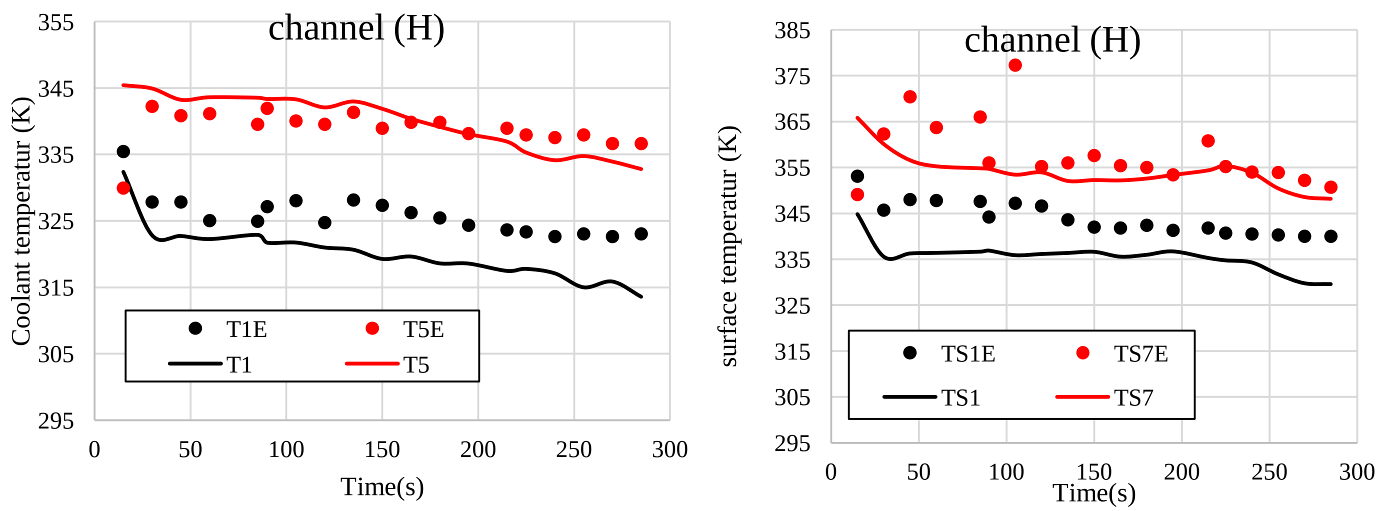
Figure (8) Comparison between experimental measurements and ANSYS results of coolant and surface temperatures at inlet and outlet of high and low power channels for right column temperature T_r=293K

4.2.1. Comparison between experimental measurements and calculated (ANSYS Model results):

Figure (9) shows the comparison at right column temperature of 303 K . there are three observations on the results. Firstly, ANSYS model predicts the occurrence of flow inversion in the channrl (L) from downward to upward, in agreement with the experimental measurments. The inversion appears in the inverse of coolant temperature at the upper and lower endsof channel (L) as figures (9a) and (9b). Secondly, ANSYS modelperdicts the increase in coolant and surface temperatures accompanying this inversion in agreement with the experimental measurments, in spite of the disagreement in the maximum values . this deviaion may be returns to the form of heat transfer coefficient in ANSYS model. Thirdly, the predicted surface temperature in (H) doesnot affected by the flow inversion in (L) in disagreement with measurments figures (9c)and (9d).



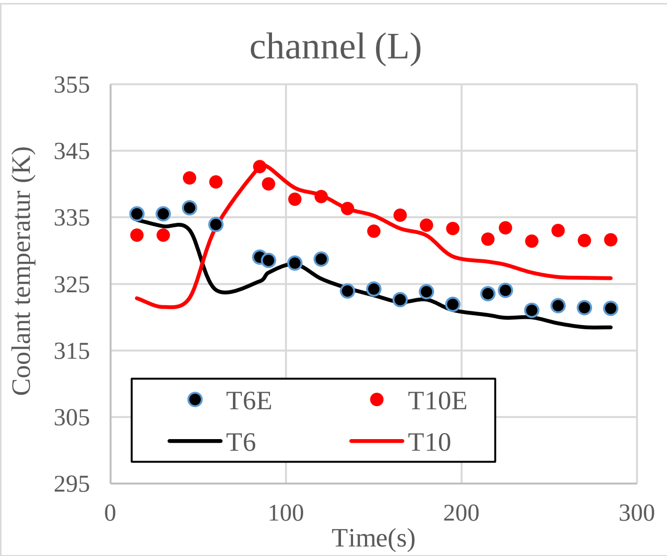
(a). (b)

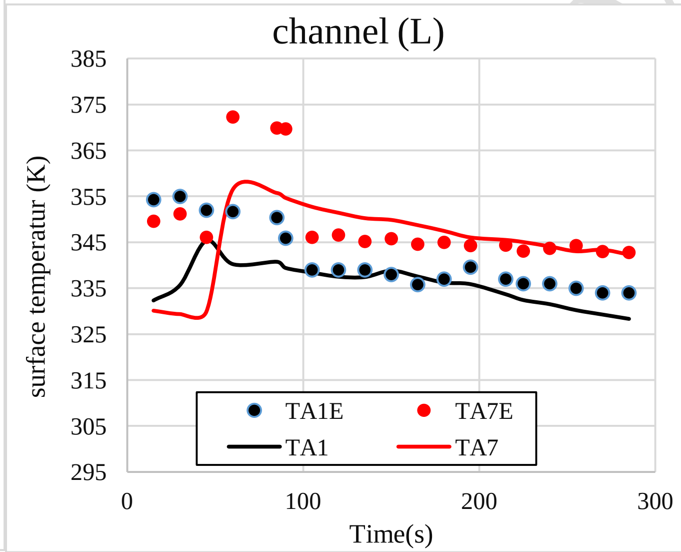


(c). (d)

Figure (9) Comparison between experimental measurements and ANSYS results of coolant and surface temperatures at inlet and outlet of high and low power channels for right column temperature T_r=303K

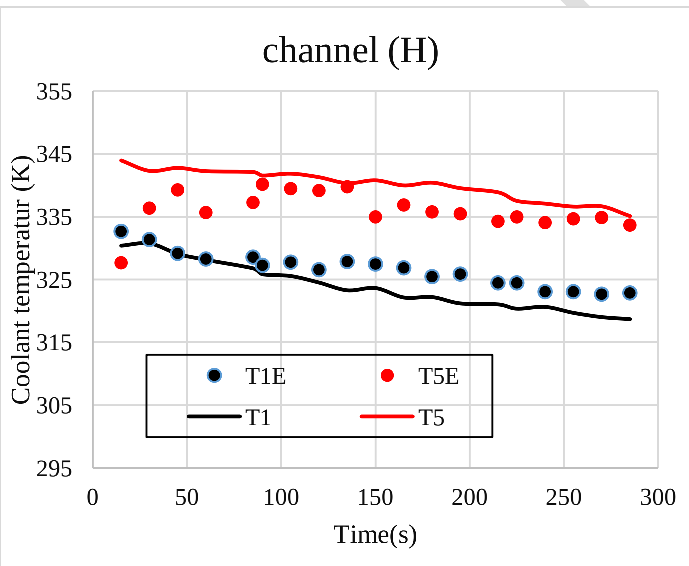
Figure (10) depicts the comparison at right column temperature of 313 K . the rsults show nearly the same qualitative behaviour likes that at 303K but with less deviation between the measurment and perdicted values. The flow inversion in (L) occurs earlier than that at 303K. also, the effect of flow inversion on the (H) which appears in the measurments as an increase in surface temperature doesnot perdicted by ANSYS model.

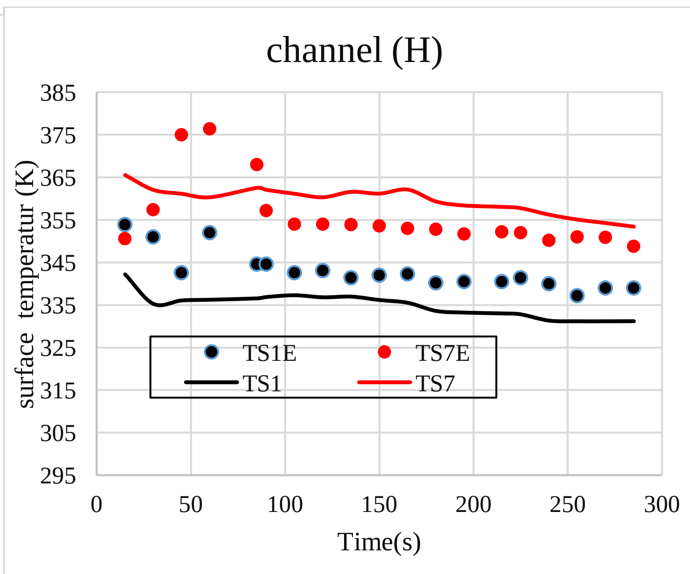




(a).

(b)





(c). (d)

Figure (10) Comparison between experimental measurements and ANSYS results of coolant and surface temperatures at inlet and outlet of high and low power channels for right column temperature T_r=313K

5.-Conclusions

The experimental investigations showed that; the new design of core cooling system in MTR research reactors has a great effect on core cooling after pump coast down. After reactor shutdown . the build up of natural circulation will depend in addition to the power distribution between the channels on the temperature difference between the reactor pool and core return pipe. At low reactor pool temperature an inner circulation between the core channels in which an upward flow in the hot channel and downward flow in the colder channels is established. When the pool temperature approaches the return pipe coolant temperatures, the internal circulation vanished. The measured temperatures showed an initial downward flow in the two channels which represents the first reverse because the simulation here for upward flow research reactor. After that, there is a sharp increase in the surface temperature appears at the occurrence time of re-reverse from downward to upward. This sharp increase in surface temperature is the most important side effect for the appearance re-reverse from the safety point of view. This is the reason beyond the regulatory requirement, which postulates that the design of upward flow research reactors must avoid the occurrence of reverse flow through the core after reactor shutdown.

COMPETING INTERESTS DISCLAIMER:

Authors have declared that no competing interests exist. The products used for this research are commonly and predominantly use products in our area of research and country. There is absolutely no conflict of interest between the authors and producers of the products because we do not intend to use these products as an avenue for any litigation but for the advancement of knowledge. Also, the research was not funded by the producing company rather it was funded by personal efforts of the authors.

References:

1. Abdel-Latif, S. H., Abdel-Hadi, E. A., Kamal Eldin, A. Talha, and Khedr, A., "Experimental Study of Core Cooling after Pump Coast-Down in MTR Pool Type Research Reactor" Ph.D Thesis, Faculty of Engineering at Shoubra, Benha University, Cairo, Egypt 2014.

2. Takeda, T., Kawamura, H., and Seki, M., **"**Natural Circulation in parallel vertical channels with different heat input", Nuclear Engineering and Design, 104, pp. 133-143, 1987.
3. Chang, H.OH and Stan, B.E.,"Critical Heat Flux for Low Flow Boiling in Vertical Uniformly Heated Thin Rectangular Channels" International Journal of Heat and Mass Transfer, Volume 36, Issue 2,pp. 325-335 March 1993.

4. Yang, .B.W., and Ouyang, W., “Dynamics and Developing of Natural Circulation Cooling from Vertical Up flow and down Flow Conditions". The 4th international Topical Meeting on Nuclear thermal hydraulics, operation and safety Columbia University, April, 1994.
5. Abe, N., Yokobori, S., Nagasaka, H., and Tsunoyama , S., “Two-Phase Flow Natural Circulation Characteristics Inside BWR Vessels”Nuclear Engineering and Design, Volume 146, Issues 1-3, , pp. 253-265 February 1994.
6. Khattab, M.S. and Mina, A. R., “Core Conversion from Rod to Plate Type Fuel Element in Research Reactors” Arab Journal of Nuclear Sciences and Applications, Volume 30, pp. 247-255, 1999.
7. Housiadas, C., “Simulation of loss-of-flow transients in research reactors ”Annals of Nuclear Energy, 27, pp.1683-1693, 2000 .
8. Wang Y. and Vafai, K., “Experimental Investigation of the Transient Characteristics on a Flat-Plate Heat Pipe during Startup and Shut down Operations” Journal of Heat Transfer Volume 122, Issue 3, pp. 525-535, August 2000.
9. ANSYS FLUENT 17.2 User’s Guide