Numerical Analysis of Critical Heat Flux Phenomenon in a Nuclear Power Plant Core Channel in the Presence of Mixing Vanes

A. Rabiee*, L. Moradi, A. Atf

School of Mechanical Engineering, Shiraz University, Shiraz, Iran

**ABSTRACT:** The necessity and importance of a high heat removal potential in various areas particularly in nuclear applications are in a direct relationship with the excessively applied heat flux level. One way to increase the heat transfer performance and subsequently enhance the threshold of the critical heat flux is to employ spacer grids accompanied by mixing vanes. In this study, the effect of the spacers with mixing vanes on the critical heat flux characteristics in the dryout condition has been numerically investigated employing the benefits of the Eulerian-Eulerian framework. In the current research, several vane angles, including vane with 0, 15 and 25 degrees in comparison with the effect of the bare spacer without any mixing vanes on the flow characteristics were examined. It was shown that the presence of the spacer alone, delays the temperature jump under critical heat flux conditions. It was also concluded that increasing the angle of the mixing vanes further improves the heat transfer performance of the system by postponing the sudden temperature jump occurring in the channel; however, the presence of the spacers and vanes in the flow field imposes an increase of the pressure drop due to the constriction on the coolant flow area.

1- Introduction

Spacer grids in the nuclear fuel rod assembly provide a mechanical support which preserves a constant distance between the fuel rods and also prevents the fuel rod damage from flow-induced vibrations. Mixing vanes attached to the spacer grids produce a swirling flow in the subchannel and consequently increase the overall heat transfer performance as an essential design factor. Enhancing the critical heat flux (CHF) threshold of boiling systems which would lead to an improved thermal efficiency and reduced operational costs, has been the main goal of researchers and scientists in various areas for several decades. Mixing vane installation is an effective technique to reach this goal. The present article covers the evaluation of the mixing vanes presence on the boiling flow field accompanied by a critical heat flux using computational fluid dynamics. The activities performed by researchers in this area are mentioned in the following.

Shin et al. [1] experimentally studied the effect of angles and positions of mixing vanes on the critical heat flux in a 2×2 rod bundle with working fluid R-134a. In the CHF experiment, various mixing vane angles between 20-40 degrees have been studied. The CHF enhancement ratio (CER) had the greatest value at 30 degrees and the maximum critical heat flux enhancement was about 19%. A CHF experiment on the position of the mixing vane also showed that by weakening the swirl flow as one of the consequences of increasing distance between grid and CHF location, CER would be decreased.

Lee et al. [2] assessed the thermal-hydraulic characteristics of hybrid mixing vanes in a 17×17 nuclear rod bundle. Different types of mixing vanes were investigated and compared with each other and it was found that the hybrid mixing vanes have a greater influence on the flow characteristics such as

**Keywords:**
- Boiling
- Critical heat flux
- Mixing vane
- Spacer grid
the effect of spacer grids on the dryout phenomenon in nuclear rod bundles representing a boiling water reactor subchannels. In the current study, air and water were considered to simulate the dryout conditions in the channel. The effect of the presence of the spacers with various shapes and also different superficial velocities on the existing thin liquid film on the heated wall was investigated. The results illustrated that the liquid film did not show a specific trend, such that it was thinned and thickened at different locations downstream of the spacer grids. Jun et al. [7] carried out a thermal hydraulic evaluation and safety analysis of a system by the TASS/SMR-S code. In the current research, the employed heat transfer model in TASS/SMR-S code was validated by the available experimental data of Bennett’s heated tube tests and THTF (thermal hydraulic test facility) [8]. The critical heat flux point and also the wall temperature along the channel were reported from the outcomes of the TASS/SMR-S code. Jayanti et al. [9] investigated the effect of spacer grids on flow and heat transfer in rod bundles. The simulation was carried out by using the computational fluid dynamics (CFD) within a Eulerian-Lagrangian framework for the calculation and consideration of various flow regimes occurring in the corresponding channel. It was shown that the single-phase convective heat transfer coefficient was increased up to 40 percent in the vicinity of the spacer grid, leading to a wall temperature reduction. In addition, some calculations regarding the droplet trajectory illustrated that the rate of droplet deposition is significantly increased in the spacer region. Nazififard et al. [10] evaluated the thermal-hydraulic characteristics of an advanced pressurized water reactor with 3×3 rod bundle array. The effect of the spacer grid and standard split-type vane was numerically investigated by the use of ANSYS FLUENT software, and it was found that the Nusselt number gradually decreased downstream of the grid spacers and mixing vane and approached to a fully developed value downstream of the spacer grids. Zhu et al. [11] investigated the effect of the spacer grid on the heat transfer of supercritical water flows in a rod bundle using commercial CFD code STAR CCM6.04. In their research, they tried to inspect the thermal-hydraulic aspect of the subchannel by considering the standard spacer grid and the spacer grid with split-vanes. It was concluded that the heat transfer performance of the channel was impressively improved within and also in the downstream of the spacer grid as a result of the reduced flow area of the channel. Seo et al. [12] experimentally and numerically studied the performance of the rotating mixing vanes in increasing the critical heat flux enhancement. A swirl generating device called the rotational vane was employed for the heat transfer characteristics enhancement purposes. The created centrifugal force as a result of employment of the moving rotating vanes would lead to a remarkable increase of the critical heat flux level. The experiments were carried out in both horizontal and vertical channels with different mixing vane shapes. Among all of the various mixing vane shapes, the moving rotational vane produced more swirl and secondary flows indicating that the best heat transfer behavior would be achieved in this circumstance. Mimouni et al. [13] predicted critical heat flux in boiling flow using computational multi-fluid dynamics. A new mechanistic model in a computational multi-fluid dynamics was suggested with the intention of simulating the wall temperature sudden jump and also the boiling crisis conditions. The achieved results were compared with the 150 tests and a relatively good agreement between the results was evidenced. Chen et al. [14] investigated the thermal-hydraulic performance of a 5×5 nuclear rod bundle with and without spacer grid. For the purpose of simulating the complicated flow in the rod bundle, various turbulence models were applied in the available CFD code. The obtained numerical results of the temperature distribution at the outlet section were compared with the experimental data as well. It was concluded that the secondary flow induced by the presence of the spacer grids has a noticeable influence on the thermal-hydraulic behavior of the system. Zhao et al. [15] carried out an experiment inspecting the heat transfer of water flowing upward in vertical annuli with spacers at high pressure conditions. The results suggest that the heat transfer coefficient would be significantly improved by the effect of the spacer on the flow field in the annuli. In addition, a correlation was proposed to involve the heat transfer improvement effects as a result of the spacers at supercritical pressure conditions.

Based on the available data in the open literature, the effect of spacer grids accompanied by the existence of mixing vanes has been extensively studied especially in boiling flow regimes. However, there are a few limited studies that present the effect of mixing vanes on postponing the dryout phenomenon when the critical heat flux conditions have been reached. Moreover, most of these studies have been performed experimentally and a minority of them are accomplished by employing computational fluid dynamics and a detailed study of the flow situation has not yet been reported. Therefore, in this study, the effect of the presence of the spacers accompanied by mixing vanes on the critical heat flux characteristics in the dryout phenomenon has been investigated numerically by employing the Eulerian-Eulerian framework for each phase. A detailed description of the current simulation algorithm, including wall heat flux partitioning, turbulence, heat, and mass transfer models is demonstrated in the following section.

2- Numerical Simulation of Boiling Flow Field

In the present study, the Reynolds averaged Navier-Stokes equations have been solved in an Eulerian-Eulerian framework for the intention of modeling the boiling flow field and also some constitutional relations have been employed to describe the boiling phenomenon in equilibrium and non-equilibrium conditions. Wall boiling phenomenon is simulated using the mechanistic Rensselaer Polytechnic Institute (RPI) boiling model and the extended formulations for the departure from nucleate boiling. With the aim of considering the wall boiling regime transition from nucleate boiling to critical heat flux, topological functions have been used. It is noted that due to the existence of the high density ratio of the liquid and vapor phases particularly in the boiling flow field, employing a suitable strategy in order to achieve a converged solution is necessary. For this reason, SIMPLE algorithm method for coupling the pressure and velocity fields which leads to a more stable solution has been used. Furthermore, to discretize the diffusion and convective terms in the governing equations, first-order upwind scheme and linear interpolation method for computing the pressure particularly on the face of every computational cell have been employed. It should be mentioned that in addition to the selection of turbulence models such as k-w, using low
under-relaxation factors particularly at the first stages of the solution process, is suitable. A comprehensive description of the models used in the available code (FLUENT) is identified in the following.

2-1- Governing equations
The governing equations, including continuity, momentum and energy equations for each phase in a generalized arrangement besides the required relations are represented as:

$$\frac{\partial (\alpha \rho V)}{\partial t} + \nabla (\alpha \rho V V) = -\nabla p + \nabla (\tau) + \sum_{r=1}^{n} \left( \dot{m}_n - \dot{m}_r \right) + s_q$$

where \( n \) is the number of phases in the system, \( \dot{m}_n \) denotes the mass transfer from \( q^a \) to \( q^b \) phase, \( \dot{m}_r \) is the mass transfer from \( q^b \) to \( q^a \) phase and \( s_q \) is the external mass source applied on the \( q^a \) phase [16].

$$\frac{\partial (\alpha \rho V V)}{\partial t} + \nabla (\alpha \rho V V V) = -\alpha \nabla p + \nabla (\tau) + \sum_{r=1}^{n} \left( \dot{m}_n - \dot{m}_r \right) + s_q$$

Subscript “\( rq \)” indicates the interaction between the two phases, \( \tau \) is the stress-strain tensor in the \( q^a \) phase, \( F^q \) is the turbulent dispersed force, \( V_q \) and \( V_r \) are the relative velocity vectors between the two phases and \( F_q, F^r \) and \( F^q \) are the external body, lift and virtual mass exchange forces, respectively [16].

$$\frac{\partial (\alpha \rho H)}{\partial t} + \nabla (\alpha \rho V H) = \tau : \Gamma ;$$

$$\nabla \alpha + \frac{\partial \alpha F}{\partial t} - \nabla q + S_{\phi,\alpha} + \sum_{r=1}^{n} \left( \dot{q}_n - \dot{m}_n H_n - \dot{m}_r H_r \right)$$

In Eq. (3), \( \dot{q}_n \) is the intensity of heat exchange between the phases, \( H_q \) and \( H_r \) are the inter-phase enthalpies [15].

2-2- Turbulence modeling
Various multiphase turbulence models, including a range of 2-equation models, are offered in the available CFD code which are all usable in boiling flows. In Eq. (4), \( \phi \) symbolizes turbulence scalar equations and consequently, the turbulence modeling equations can be represented by the following general equation:

$$\frac{\partial (\alpha \rho \phi)}{\partial t} + \nabla (\alpha \rho V \phi) = \nabla (\alpha \Gamma \nabla \phi) + \alpha S_{\phi,\phi}$$

In this equation, \( \Gamma_{\phi,\phi} \), \( S_{\phi,\phi} \) are the diffusion coefficient and the general source term which include production, dissipation and extra source terms stemming from turbulent-bubble interactions [16]. It is noted that k-\( \varepsilon \) turbulence model has been used in this research, preferably.

$$\frac{\partial (\alpha \rho k)}{\partial t} + \nabla (\alpha \rho V k) = \nabla (\alpha \Gamma_{k,k} \nabla k) + \alpha G_{k,\phi} - \alpha \rho \varepsilon + \alpha \rho \varepsilon \prod_{q} \left( k_q \right)$$

2-3- Wall boiling models
RPI model developed by Kurul and Podowski [17] has been widely used as the modeling approach for the mechanistic prediction of boiling phenomenon. It is notable that in the current study, the applied total wall heat flux is divided into three parts, namely liquid phase convective heat flux \( q_c \), quenching heat flux \( q_q \) and evaporation heat flux \( q_e \) as shown in Fig. 1.

$$\dot{q}_w = \dot{q}_c + \dot{q}_q + \dot{q}_e$$

2-4- Liquid phase convective heat flux
In Eq. (8), \( h_l \) is denoted as the liquid phase heat transfer coefficient, \( T_w \) and \( T_i \) are the wall and liquid temperature near the wall, respectively [16]. \( A_s \) is the area of influence which could be calculated in terms of the bubble departure diameter and the nucleate site density as follows:

$$A_s = \min \left( \frac{1}{\eta}, \frac{Z}{d} \right) \frac{2 \pi}{A}$$

The empirical coefficient \( \eta \) can be calculated by the relation suggested by Valle and Kenning in 1985 [18]:

$$\eta = 4.8 \exp \left( - \frac{Ja}{80} \right)$$

And \( Ja \) is the subcooled Jacob number, which is defined as:

$$Ja = \frac{\rho C p \Delta T_{nb}}{\rho l H_w}$$

2-5- Quenching heat flux
Quenching heat flux partition is expressed as the following relation:

$$\dot{q}_q = C_{\text{wq}} \frac{2k}{V_{\text{wq}}} (T_w - T_{\text{wb}}) A_s$$
This term simulates the cyclic averaged transient energy transfer associated with liquid filling the wall vicinity after the bubble is detached from the wall with a period of $T_{cw}$ [16]. $k_p$, $y$, and $f_{cw}$ are heat conductivity, diffusivity in the liquid phase and the frequency of bubble departure, respectively and $C_{nw}$ is a constant coefficient, as well.

2-6- Frequency of bubble departure
In this research, the frequency of bubble departure is computed based on the inertia controlled growth proposed by Cole [19]:

$$f_{cw} = \sqrt{\frac{4g(\rho_v - \rho_l)}{3\rho_d d_{cw}}}$$  \hspace{1cm} (13)

2-7- Evaporation heat flux
Evaporation heat flux is calculated by the following equation:

$$\dot{q}_v = \frac{\pi^2 d_{cw}^4}{6} f_{cw} N_v \rho_v H_v$$  \hspace{1cm} (14)

Where $d_{cw}$ is the bubble departure diameter, $N_v$ is the active nucleation site density, $\rho_v$ is the vapor density, and $H_v$ is the latent heat of evaporation.

2-8- Nucleation site density
In this study, the magnitude of nucleation site density is represented by the Lemmert-Chawla correlation in terms of the wall superheat [20]:

$$N_v = C^* (T_{cw} - T_{sw})$$  \hspace{1cm} (15)

Here by default, $C^*$ = 15545.54 and $n$ = 1.805.

2-9- Bubble departure diameter
Bubble departure diameter is modeled by the correlation proposed by Tolubinski [21]:

$$D_{dw} = \min(0.0014, 0.0006 \exp(-\frac{AT_{cw}}{45}))$$  \hspace{1cm} (16)

The function $f(\alpha_v)$ is in terms of the local vapor volume. It can be predicted by the relation offered by Ioilev et al. [22]:

$$f(\alpha_v) = 1 - f(\alpha_v) = \max(0, \min(1, \frac{\alpha_v - \alpha_{v,1}}{\alpha_{v,2} - \alpha_{v,1}}))$$  \hspace{1cm} (17)

2-10- Modified RPI wall boiling model
In order to model the critical heat flux phenomenon, it is crucial to include the vapor temperature variation in the governing equations, while in the conventional RPI model, the vapor temperature is not calculated from solving the phase energy equation but fixed at the saturation temperature. In addition, thin liquid films along heated walls also need to be taken into account in the wall heat flux partitioning equation. For that reason, the wall heat partition is modified as follows [22]:

$$\dot{q}_w = (\dot{q}_c + \dot{q}_h + \dot{q}_b + \dot{q}_v) f(\alpha_v) + (1 - f(\alpha_v)) \dot{q}_c + \dot{q}_h$$  \hspace{1cm} (18)

The function $f(\alpha_v)$ is in terms of the local vapor volume. It can be predicted by the relation offered by Ioilev et al. [23, 24]:

$$f(\alpha_v) = 1 - f(\alpha_v) = \max(0, \min(1, \frac{\alpha_v - \alpha_{v,1}}{\alpha_{v,2} - \alpha_{v,1}}))$$  \hspace{1cm} (19)

It is noteworthy to mention that the flow regime transition occurs smoothly from a continuous liquid bubbly flow to churn and finally a continuous mist flow configuration in terms of single local flow quantity named the vapor volume fraction, $\alpha_v$ as follows [16, 22].

Bubbly flow topology: the vapor phase is dispersed in the continuous liquid in the form of bubbles ($\alpha_v \geq 0.3$). Churn flow: this is an intermediate topology between the bubbly and mist flow topology, where $0.3 \geq \alpha_v \leq 0.7$. Mist flow topology: the liquid phase is dispersed in the continuous vapor in the form of droplets $\alpha_v \geq 0.7$.

Interfacial quantities such as interfacial area, drag, lift, turbulent drift force and heat transfer named $\omega$, can be computed using the following general form:

$$\omega = (1 - f(\alpha_v)) \omega_{bubbly} + f(\alpha_v) \omega_{turbulent}$$  \hspace{1cm} (20)

Here $\omega_{bubbly}$ and $\omega_{turbulent}$ are the interfacial quantities from the bubbly flow and mist flow, respectively [16].

2-11- Interfacial transfer
In this study, the interfacial area, momentum transfer terms such as drag, lift, and turbulent quantities are determined as follows:

2-11-1- Interfacial area
The interfacial area is one of the important parameters for the momentum and heat transfer processes. The following relation is used in the current study:

$$A_v = \frac{6(1 - \alpha_v)\alpha_v}{D_{ew}}$$  \hspace{1cm} (21)

Here, subscript “d” represents a dispersed phase, which is the vapor phase in bubbly flows and the liquid phase in mist flows [16].

2-11-2- Interfacial momentum transfer
In boiling flow regimes, the most important interfacial momentum transfers are drag, lift, and turbulent drift forces which are described in the subsequent sections.

2-11-3- Interfacial drag force
For dispersed bubbly and mist flows, the interfacial drag force has the general form:

$$F^D = -F^D_{ew} = \frac{A_v}{8} \rho_l C_{h,v} (V_1 - V_2) \left| V_1 - V_2 \right|$$  \hspace{1cm} (22)

Where subscript “e” indicates a continuous phase, $\rho_l$ is the
liquid density in the bubbly regime, but the vapor density in the mist regime. The drag coefficient $C_{lv}$ can be computed by various models [16].

2- 11- 4-  Interfacial lift force
The interfacial lift force has the general form:

$$F_i^l = -F_i^l = -C_{lv} \rho_l \alpha_d \nabla \times (V \times V_c)$$

(23)

Where $\alpha_d$ is the volume fraction of a dispersed phase. $V_c$ is the velocity vector of a continuous phase. The coefficient, $C_{lv}$ can be calculated by Moraga et al. or Tomiyama et al. formulations [23] in bubbly flow regimes, while it is assumed to be zero in droplet flows.

2- 11- 5-  Turbulent drift force
In the RPI boiling model, the turbulent drift force is computed as [24]:

$$F_{br}^{TD} = -F_{vl}^{TD} = -C_{TD} \rho_c k_e \nabla \alpha_d$$

(24)

Here $k_e$ is the turbulent kinetic energy of the continuous phase. The turbulent disperse coefficient $C_{TD}$, by default, is set to 1.

2- 12-  Modified RPI wall boiling model
In order to model the critical heat flux phenomenon, it is crucial to include the vapor temperature variation in the governing equations, while in the conventional RPI model, the vapor temperature is not calculated by solving the phase energy equation but is fixed at the saturation temperature. In addition, thin liquid films along heated walls also need to be taken into account in the wall heat flux partitioning equation. For that reason, the wall heat partition is modified as follows [22]:

$$\dot{q}_w = \dot{q}_l + \dot{q}_{lv} + \dot{q}_j + \dot{q}_r f(\alpha_v) + (1 - f(\alpha_v))\dot{q}_l + \dot{q}_j$$

(25)

The function $f(\alpha_v)$ is in terms of the local vapor volume. It can be predicted by the relation offered by Ioilev et al. [23, 24]:

$$f(\alpha_v) = 1 - f(\alpha_v) = \max(0, \min\left(1, \frac{\alpha_v - \alpha_{crit}}{\alpha_{crit} - \alpha_{crit}}\right))$$

(26)

3- Submission of Papers to Journal
This section presents some results which include the modeling of single-phase flow field in the normal conditions in order to investigate the mixing vanes operation and adjusting the available code. In the next step, the effectiveness of mixing vane installation under accident conditions in a boiling flow field accompanied by dryout phenomenon has been examined eventually.

3- 1- Single and two-phase flow boiling
The first validation case is a 3D PWR core channel as shown in Fig. 2. In this figure, a schematic of the 5×5 fuel rod bundle and mixing vane geometry attached to the spacer grid with an angle of 25 degrees accompanied by its arrangement have been illustrated. For further detailed information, one could refer to reference [4, 25]. The inlet boundary conditions considered in the present simulation correspond to the experiments performed by Karoutas et al. [25], with temperature and operating pressure set to 26.67 °C and 4.83 bar, respectively. An axial inlet velocity equal to 6.79 m/s was defined at the entrance of the bundle.

Figs. 3 and 4 depict the computational model with the associated grids and a schematic of the extended region of the mixing vanes attached to the spacer grids. It is noted that due to the imposed symmetric boundary condition, the extended computational domain could be developed as shown in the left-hand side of the Fig. 4. In order to investigate the grid independency of the solution, unstructured grids with 923169 and 1296116 cells, were employed and the two grids returned almost similar solutions.

Figs. 5 and 6 present a comparison between the computations and the experimental data consisting of static pressure along the channel and lateral velocity at section 0.1905 meters from the inlet. As can be seen from these figures, the results of the current study along with the Karatous and In et al. computations [25, 26] are in a close agreement with the experimental data.
As an introduction to approach the critical heat flux phenomenon, the boiling case of the current study by applying a heat flux equal to 5000 kW/m$^2$ has been investigated. Fig. 7 represents the boiling heat transfer coefficient. As can be seen from this figure, the presence of the spacer grid has led to a sharp increase in the local heat transfer coefficient.

### Critical heat flux results

In the current section, the results of the critical heat flux phenomenon due to the excessive heat are reported in a vertical rod bundle. It is notable that the effects of mixing vanes are discussed subsequently. Experiments were performed in the THTF (Thermal-Hydraulic Test Facility), carried out by the ORNL (Oak Ridge National Laboratory) [27]. THTF is nonnuclear pressurized water loop containing 64 full-length rods (fuel rod simulator) arranged in an 8 × 8 bundle. Rod diameter (0.0095 m) and pitch (0.0127 m) are typical of a PWR with 17 × 17 fuel assemblies (Fig. 8). The axial and radial power distributions of the THTF bundle are flat. The heated length and diameter of the bundle are 3.66 m and 0.95 cm, respectively. There are six spacer grids in the heated length. At the inlet, water enters the channels with subcooling temperature equal to 19.1 K, the saturation temperature of 602.56 K and the wall heat flux on the heated rods is 0.914 MW/m$^2$. Liquid flows upward through the subchannels with a mass flux of 705 kg/m$^2$s under operating pressure of 12.76 MPa. Figs. 9 and 10 depict a schematic of the corresponding channel with boundary conditions and the associated computational grids, respectively.

It is observed that the amount of 823310, 835820 and 835820 computational cells for the cases accompanied by mixing vane of 0, 15 and 25 degrees have been used, respectively. Fig. 11 represents the wall temperature distribution with and without the spacer grids besides experimental data [27] together with Jun et al. [7]. It can be seen that a jump in the wall temperature with an average magnitude approximately equal to 1000 K has occurred in the vicinity of 1.6 m from the inlet according to experimental data. As can be observed, the temperature profiles are in a relatively good agreement.
with the experiment and a better solution has been achieved compared to the Jun et al.’s result, although the current study predicts the temperature jump location earlier. It is noteworthy to say that the presence of the spacer grids predicts the temperature jump location better compared to the experimental data; however, it slightly underestimates wall temperature in the post-CHF region except for the near exit region which is in a good agreement with the test data. It is also evident that in the case of the spacer grids, there exist some fluctuations in the wall temperature as a result of the spacer grids presence particularly after the occurrence of the temperature jump.

3- 2- 1- CHF and post-dryout in rod bundle in the presence of mixing vanes
As illustrated in the preceding section, post-CHF heat transfer regimes are inefficient and its occurrence can cause a large temperature gradient in the heated wall, leading to physical burnout. In this section, it is tried to enhance the critical heat flux threshold by adding mixing vanes to different angles. It is noteworthy to mention that the geometric configuration and boundary conditions are the same as previous section 2.2. The problem is treated as a steady-state, liquid-vapor two-phase turbulent boiling flow of water with the saturation temperature of 602.56 K.

The influence of mixing vanes with different angles attached to the spacer grids on thermal-hydraulic properties is represented in Fig. 12. It can be seen that sudden changes in the wall temperature will be delayed in the presence of the mixing vanes with different angles. It is noteworthy to mention that increasing the angle of the mixing vanes from 0 to 25 degrees, postpones the sudden temperature rise and also decreases the maximum temperature value.

Fig. 12. Wall temperature variation with different vane angles

Fig. 13. Surface heat transfer coefficients along the channel

The average heat transfer coefficients for different cases have been computed and gathered in Table 1. As can be seen, the
presence of vane with an angle of 25 degrees would enhance the heat transfer coefficient to approximately 40 percent. Description of velocity vectors at a section after the vane with 25 degrees angle has been depicted below in Fig. 14. As can be seen from this figure, the presence of vane leads to the rotational flow after the van and consequently increases the turbulent intensity and heat transfer capacity.

Fig. 15 represents the vapor volume fraction along the channel in the presence of the spacer grids with different mixing vane angles. Despite the sudden change in the wall temperature, the vapor volume fraction has increased gradually along the channel. As can be seen, increasing the vane angle reduced the rate of vapor formation particularly in the vicinity of the entrance region.

Fig. 16 depicts the wall heat partitioning for the mixing vane with 25 degrees as a typical case. The sum of the different wall heat flux partitions is equal to the total heat flux (900000 W/m²). It is evident that up to the length 2.5 m from the channel inlet, almost all of the heat flux applied to the wall is allocated to the evaporation heat flux contribution while at downstream of the post-dryout, vapor heat flux plays a dominant role. It is notable that all of the following results correspond to the mixing vane with 25 degrees.

Fig. 17 shows the water volume fraction contours at several selected sections along the channel. It could be seen that water begins to evaporate immediately after entering the heated channel and the vapor volume fraction steadily increases along the axial direction.

Fig. 18 and 19 illustrate the water volume fractions at different sections before and after the mixing vane and also for the case without mixing vane. Based on these figures and Fig. 15 in the manuscript, the presence of the vanes due to the enhanced turbulent intensity, the amount of the void fraction along the wall channel has been decreased.

A description of the temperature contours has been depicted in Fig. 20. As can be seen, the near-wall liquid layer has been highly affected by the heated wall in comparison with other regions.

Figs. (21-24) show liquid and vapor velocities at the sections, 0.25, 0.335, 2 and 3.65 meters before and after the occurrence of the dryout along the channel and at the outlet, as well.
Due to the departure from nucleate boiling occurrence, it is evident that a significant increase in the speed of the phases has happened. It is also seen that at identical sections, the liquid and vapor velocity distributions are not equal, which suggests that there exists a slip velocity between the two phases. It is notable that due to the swirling flow as a result of the existence of mixing vanes, there is an asymmetry in the lateral velocity profiles, however, as the flow approaches the exit region, this asymmetry would be diminished.
4- Conclusion

In this research, efforts have been made to examine the effects of the mixing vanes attached to the spacer grids on the flow field in CHF conditions. In order to simulate the boiling flow field, averaged Navier-Stokes equations have been used with a Eulerian- Eulerian approach for each phase and wall boiling phenomenon is modeled using the mechanistic RPI model and the extended formulations for the departure from nucleate boiling, as well. In the first step, a 5×5 PWR fuel rod bundle was simulated in order to adjust the available code and it was observed that a fairly good agreement was achieved between the results and the experimental data. The main objective of this study was to investigate the critical heat flux phenomenon in an 8×8 rod bundle in the existence of spacer grids accompanied by the mixing vanes with different angles on the deviation from nucleate boiling. According to the obtained results, the presence of the spacer grid alone in the flow field postpones the occurrence of the dryout phenomenon to about 0.25 meters (Fig. 11) and also increasing the vane angle to 25 degrees delays the sudden temperature rise approximately about 1 meter (Fig. 12). In addition, attaching the mixing vanes to the spacer grid also improves the heat transfer characteristics of the flow field such that it delays the sudden temperature jump occurrence and also decreases the maximum temperature value about 100 degrees (Fig. 12). Moreover, it was concluded that increasing the mixing vane angle from 0 to 25 degrees further promotes the heat transfer coefficient of the system about 40 percent based on Table 1. It can be summarized that the presence of mixing vanes in the flow field enhances the heat transfer performance of the system and consequently improves the safety margins leading to a secure operation of the nuclear reactor in both normal and accident conditions.

References


